

AFT Fathom[™]

Help Guide with Examples

Metric Units

AFT Fathom Version 13

Incompressible Pipe Flow Modeling

Published: February 10, 2025



Dynamic solutions for a fluid world[™]

Contents

Overview of AFT Fathom	27
Engineering Assumptions	28
New in Version 13	29
Interface Improvements for Fathom 12	38
New in Version 12	40
New in Version 11	43
New in Version 10	48
New in Version 9	54
FAQ - Frequently Asked Questions	56
Fathom Learning Resources	63
Startup Window	64
Fundamental Modeling Concepts	68
Pipes and Junctions	68
Reference Positive Flow Direction	69
Features for Modeling Irrecoverable Losses	70
Philosophy of Computer Modeling	70
Role of Pressure Junctions and How They Work	72
Open vs. Closed Systems	75
Curve Fitting	79
Stagnation vs. Static Pressure Boundaries	80
Analysis Setup Overview	83
Modules Group	84
Modules Panel	85
Fluid Properties Group	90

Fluid Panel	91
Viscosity Model Panel	97
Heat Transfer/Variable Fluids Panel	120
Laminar and Non-Newtonian Corrections Panel	123
SSL Module Fluid Properties Group	131
Pipes and Junctions Group	141
Pipes and Junctions Panel	141
Transient Control Group	143
Simulation Duration Panel	143
Goal Seek and Control Group	148
Variables Panel	149
Goals Panel	155
Numerical Controls Panel	161
Steady Solution Control Group	162
Tolerance Panel	163
Iterations Panel	165
Temperature Limits Panel	167
Environmental Properties Group	168
Environmental Properties Panel	169
Cost Settings Group	170
Cost Settings Panel	171
Automatic Sizing Group	173
Sizing Summary Panel	174
Miscellaneous Group	175
NFPA Panel	175
Heat Transfer Sectioning Panel	178

Advanced Parameters Panel	179
Working with Pipe and Junction Objects	181
Selecting and Finding Objects	181
Selecting Objects	181
Finding Objects	184
Groups	185
Pipe Drawing Modes	186
Global Edit	193
Global Pipe Editing	193
Global Junction Editing	193
Toolbox	195
Creating Pipes - the Pipe Drawing Tool	197
Joining or Splitting Pipes	199
Creating Junctions	202
Connecting Objects	203
Moving Objects	204
Editing Objects	205
Morphing Junctions	206
Defining Objects	206
Highlight Required Input	208
Inspecting Objects	209
Notes	210
Status	211
Workspace Pan	211
Moving Pipe and Junction Labels	212
Special Conditions	213

Pipes	214
Pipe Properties Window	214
Common Pipe Input Parameters	215
Pipe Model	216
Fittings & Losses	218
Heat Transfer in Pipes	220
Constant Heat Transfer	223
Convective Heat Transfer	224
Buried Pipe Heat Transfer	227
Heat Tracing Flux based on Wire Turns and Length	227
Heat Transfer in Pipes - Detailed Discussion	231
Insulation for Pipes	237
Fluid Properties for Pipes	238
Optional Input for Pipes	238
Add/Remove Pipe Segment	241
Reverse Direction	242
Pressure Drop in Pipes - Detailed Discussion	243
Junctions	248
General Junction Information	249
Two Window Formats	251
Common Junction Input Parameters	254
Optional Input for Junctions	255
Specifying Losses	257
Conventions for Specifying Basis Area	257
Reference Density	258
Equal Percentage Valves	260

Junction Loss Models	261
K Factor Loss Model	261
Kv and Cv Loss Model	263
Resistance Curve Loss Model	264
3-K (Darby) Method	266
Equivalent Length	267
Area Change	269
Area Change Loss Model	269
Assigned Flow	271
Assigned Pressure	272
Bend	272
Bend Loss Model	274
Branch	277
Check Valve	277
Control Valve	278
Control Valve Setpoint Not Achievable	280
Dead End	281
General Component	282
Heat Exchanger	282
Heat Exchanger Heat Transfer	283
Jet Pump	287
Orifice	287
Orifice Loss Model	288
Pump	291
Centrifugal (Rotodynamic) Pump	294
Positive Displacement Pump	314

Pump as Turbine Overview	317
Affinity Laws	324
Submerged Pump	326
Pump Efficiency Data	328
Relief Valve	330
Reservoir	333
Screen	335
Screen Loss Model	336
Spray Discharge	337
Spray Discharge Fire Sprinklers	339
Spray Discharge CD (Discharge Coefficient)	340
Tee/Wye	341
Tee/Wye Loss Model	342
Three Way Valve	349
Valve	350
Valve Modeling Closures with K Factors	352
Venturi	352
Volume Balance	353
Weir	355
Interface and User Options	359
Menus and Toolbars	359
File Menu	360
Edit Menu	361
View Menu	363
Analysis Menu	363
Tools Menu	364

Library Menu	365
Arrange Menu	366
Window Menu	368
Help Menu	368
Toolbars	369
Printing	370
Primary Window Overview	372
Status Bar	373
Quick Access Panel	374
Model Status Light	383
Solution Progress Window	383
Workspace Window	384
Workspace Layers	385
Annotations	395
Renumbering Workspace Objects	399
Merging Models	400
Scale/Flip Workspace	401
Magnifier Tool	401
Duplicate Special	402
Undefined Objects Window	403
Keyboard Shortcuts for Workspace	404
Extended Model Check	406
Raw Data Changes	407
Model Data Window	408
Model Data Control	410
Bill of Materials	412

Output Window	413
Output Control	418
Output Parameters	424
Applied Standards	426
Graph Results Window	426
Quick Graph Formatting Window	432
Graph Types	432
Pump vs. System Curve Methods	435
Basic Graph Tutorial	436
Multi-Scenario Pump vs. System Curves Tutorial	440
User Options Overview	446
Unit System	447
Workspace	449
Parameters	452
Files	454
Sizing Preferences	456
File Import and Export	458
Importing Piping Component Files	458
Importing GIS Shapefiles	462
Importing CAESAR II Neutral Files	464
Importing and Exporting EPANET Files	467
Importing and Exporting from Other AFT Products	471
Exporting To Excel	472
Importing From Excel	478
Special Modeling Features	490
Submergence	490

Design Alerts	491
Design Alerts From Output	494
Design Alerts Importing & Exporting	496
Scenario Manager	498
Creating, Organizing, and Editing Scenarios	500
Modifying Individual Scenarios	503
Scenario Inheritance	504
Running Models in Batch Mode	508
Open Model Using Last Scenario	509
Disconnected Scenario	510
Scenario Comparison Tool Overview	510
Accessing Scenario Comparison Tool	511
Modify Comparison	512
Scenario Comparison Grid	513
Cost Calculation	516
Enabling Cost Calculation	517
Energy Cost Library	518
Cost Application Manager	521
Scale Tables	523
Select Size From	525
Cost Definitions	526
Cost Libraries	529
Cost Summary Window	533
Custom Monetary Units	534
Cost Report	535
Add-on Modules	538

GSC - Goal Seek & Control	538
GSC in Model Data	540
GSC in Solution Progress	542
GSC in Output	543
GSC in Scenarios	543
Changing Input Values to GSC Results	544
GSC Troubleshooting	545
SSL - Settling Slurry	550
Slurry Calculation Method Overview	552
SSL Nomenclature	570
SSL Engineering Assumptions	571
SSL Flow Rates	572
Estimating M From Recorded Data	573
Solids Library	574
SSL Pump De-Rating	577
SSL in Output	580
SSL in Graph Results	581
SSL Initial Guesses for Pipe Velocity	583
SSL Troubleshooting	584
Tips When Using GSC with SSL Module	585
SSL Pipe Slope	585
XTS - Extended Time Simulation	585
Choosing Time Simulation	588
Junction Transient Data	588
Finite and Infinite Reservoirs	601
XTS in Workspace	610

XTS in Model Data	611
XTS in Output	612
XTS in Graph Results	613
ANS - Automated Network Sizing	620
Sizing Fundamentals	622
Sizing Window	636
Solution Progress with ANS	659
Sizing Results	660
Workspace Integration	662
Junction Loss Factors	664
Improving and Troubleshooting the Sizing Process	664
Sizing Components Other Than Pipes	666
Search Methods	678
Sizing Theory	688
Libraries	707
Library Manager	708
Library Browser	708
Junction Library	712
Fluid Library	713
Pipe Material Library	714
Insulation Library	717
Fittings & Losses Library	717
Network Library Overview	723
Benefits of Shared Libraries	723
Creating an Enterprise-Wide Network Library System	724
Library Verification	727

Library Locations in General Preferences	729
Theory and References	731
Nomenclature	731
Network Solution Methodology	733
Continuity Equation at a Branch	737
Energy Equation at a Branch	737
Network Implementation	737
Verifying Network Solutions	740
References	741
Chempak Technical Information - Detailed Discussion	743
Role of Pressure Junctions - Detailed Discussion	755
Pipe Material Library References	766
Output Messages & Warnings	772
Critical Warnings	772
FCV/PRV/PSV could not control pressure - Pressure was added to achieve control flow/pressure	773
Global energy balance disabled by user in Advanced Parameters Panel	774
Global mass balance disabled by user in Advanced Parameters Panel ..	774
Junction X had reverse flow - Pump head could not be predicted	774
Pipe sequences of zero friction exist, flow rate cannot be calculated	774
Setting relaxation lower than relative pressure/flow tolerance gives unre- liable results	774
There were static pressures below absolute zero in the model	775
These results do not represent a converged solution	776
Weir Liquid Level Convergence Problem	776
Warnings	776
Cavitation May Exist at Junction X	776

Chempak temperature and or pressure is out of range for the following ..	776
Control valve open percentage outside valid range of 0-100% at Junction X	777
Energy may not be balanced at Junction X (total= Y)	777
FCV could not control flow at Junction X	777
Flow is into system at Reservoir Junction X	777
Flow may not be balanced at Junction X (total= Y)	778
Flow rate at drop off point at Junction X	778
GSC Goal may not be valid. User goal = X. Final goal result = Y.	778
HGL, EGL and head loss results may not be meaningful for variable density systems	779
Junction X %BEP (Y%) is above/below the POR range	779
Junction X exceeded runout flow rate	779
Junction X flow (Y gal/min) is above/below the AOR range (Z gal/min)	779
Junction X has insufficient pump NPSH/NPSP... ..	779
Negative added pressure by pump at Junction X	779
Pipe X has an angle of Y degrees, but the Wasp-Durand model does not account for inclined or vertical pipes	780
Pipe X temperature is equal to or greater/less than the maximum/minimum temperature limit of... ..	780
PRV could not control pressure at Junction X	781
PSV could not control pressure at Junction X	781
Pump X had efficiency greater than 100%	781
Static pressure lower than vapor pressure at inlet/outlet of pipe/junction X	782
Tank Junction X began overflowing at Y seconds	782
Tank Junction X completely drained at Y seconds	782
Temperature at inlet/exit of pipe X is above/below max/min fluid temperature range	782

Temperature is above the saturation line for the following...	783
Unique solution not possible - Relief valve cycles	783
Velocity in pipe X is less than the deposition velocity	784
Cautions	784
Check Valve closed at Junction X	785
Default control valve actions at Junction X used when setpoint not achievable because analysis is transient	785
Flow is negative through Junction X. Loss factors may be in error.	785
Flow solution for Junction X is above/below the maximum/minimum value specified for the resistance curve. Flow rate: Y	785
Head loss, head rise, hydraulic gradeline and energy gradeline output does not have meaning for zero g systems	786
Head rise across fixed flow pumps is referenced to system g level...	786
Junction X has a density change across it	786
Losses at one or more pipes or junctions may be significantly impacted (>25%) by laminar or non-Newtonian conditions, but no correction was made	787
Note: No scenarios have been set in the Multi-Scenario Output display	788
Pipe X elevation of Y is below Reservoir Z tank bottom elevation	788
Pipe X has an angle of Y degrees, but the Wasp-Durand model does not account for inclined or vertical pipes	788
Pipe X length is shorter than the elevation change	789
Pipe X segment length Y is shorter than the elevation change	789
Pipe X steady state Reynold's Number is less than 100. Minor Losses may be significantly lower than expected.	789
Pump X has no efficiency curve data. Nominal efficiency was used in viscosity correction calculations	789
Pump X has no efficiency curve or nominal efficiency data...	790
Pump X properties were outside of the valid range for the viscosity correction calculations. Results may be invalid.	790

Spray Discharge Junction X closed because inflow occurred and Special Condition - No Inflow was set	790
System Life specified in Cost Settings is different from the XTS time	790
The following Detailed Tee/Wye junctions have a different automatic orientation than defined on the Workspace	790
The performance of Pump Junction X may be affected by the fluid viscosity	792
Transient data did not extend to Stop Time at Junction X	792
Wasp-Durand relative concentration (X) exceeds 0.8	792
Information	793
Steady-state conditions are laminar in the following pipes	793
Troubleshooting	797
Diagnostics Window	797
Use the Model Data Window	798
Transfer Results to Initial Guesses	800
Poor Pump Curve Fits	800
Troubleshooting Resistance Curves	801
Use the Output Window Sort Feature	802
Tee/Wye Junction Complexity	803
Lower the Flow Rate Relaxation	803
Try Absolute Tolerance	804
Make Initial Flow Rate Guesses for Pipes	804
Review the Iteration History	804
Change Boundary Conditions	805
Turn Off Parts of the Model	805
Break a Large Model Into Submodels	806
Identifying and Restoring Corrupted Model Files	807

Technical Support	811
Technical Support Options	811
Contacting AFT	812
Information on Software Installation	813
Chempak Database Installation	813
Output Files	815
Updated Fluids Library	815
Special File Control Features	816
Converting Models from Earlier Versions	817
Examples	819
Walk-Through Examples	819
Beginner - Three Reservoir	823
Sizing a Pump	848
Control Valve	856
Heat Transfer in a Pipe	861
Heat Exchanger System	866
Spray Discharge System	874
Pump Sizing and Selection with Flow Control Valves	882
Freon Delivery System with Heat Transfer	888
Cooling System	895
Housing Project	900
Beginner - Cost Calculation	908
Non-Newtonian Phosphates Pumping	918
NFPA Firewater System	934
Crude Oil Pipeline	943
Hot Water System	950

Plant Cooling Cost Calculation	958
Non-Newtonian Mill Discharge Slurry	974
Flow Induced Vibration Calculations (Metric Only)	985
Module Walk-Through Examples	992
Goal Seek and Control (GSC) Examples	993
Extended Time Simulation (XTS) Examples	1037
Settling Slurry (SSL) Examples	1091
Automated Network Sizing (ANS) Examples	1133
Combined Modules Examples	1263
Verification	1277
AFT Fathom Verification Overview	1277
References	1278
Verification Methodology	1279
Verification Methodology	1279
ANS Module Verification Methodology	1280
Verification Models	1281
Summary of All Verification Models	1281
Summary of Verification Models with Pumps	1286
Summary of Verification Models with NPSH	1288
Summary of Verification Models with PRVs	1289
Summary of Verification Models with Slurries	1290
Summary of Verification Models with ANS	1291
Verification Cases	1292
Verification Case 1 Problem Statement	1292
View Verification Case 1 Model	1294
Verification Case 2 Problem Statement	1297

View Verification Case 2 Model	1301
Verification Case 3 Problem Statement	1303
View Verification Case 3 Model	1305
Verification Case 4 Problem Statement	1307
View Verification Case 4 Model	1310
Verification Case 5 Problem Statement	1312
View Verification Case 5 Model	1314
Verification Case 6 Problem Statement	1316
View Verification Case 6 Model	1317
Verification Case 7 Problem Statement	1319
View Verification Case 7 Model	1320
Verification Case 8 Problem Statement	1322
View Verification Case 8 Model	1325
Verification Case 9 Problem Statement	1327
View Verification Case 9 Model	1330
Verification Case 10 Problem Statement	1332
View Verification Case 10 Model	1334
Verification Case 11 Problem Statement	1336
View Verification Case 11 Model	1338
Verification Case 12 Problem Statement	1340
View Verification Case 12 Model	1341
Verification Case 13 Problem Statement	1343
View Verification Case 13 Model	1345
Verification Case 14 Problem Statement	1347
View Verification Case 14 Model	1348
Verification Case 15 Problem Statement	1350

View Verification Case 15 Model	1351
Verification Case 16 Problem Statement	1353
View Verification Case 16 Model	1355
Verification Case 17 Problem Statement	1357
View Verification Case 17 Model	1359
Verification Case 18 Problem Statement	1361
View Verification Case 18 Model	1364
Verification Case 19 Problem Statement	1366
View Verification Case 19 Model	1369
Verification Case 20 Problem Statement	1371
View Verification Case 20 Model	1372
Verification Case 21 Problem Statement	1374
View Verification Case 21 Model	1375
Verification Case 22 Problem Statement	1377
View Verification Case 22 Model	1379
Verification Case 23 Problem Statement	1382
View Verification Case 23 Model	1384
Verification Case 24 Problem Statement	1386
View Verification Case 24 Model	1388
Verification Case 25 Problem Statement	1390
View Verification Case 25 Model	1391
Verification Case 26 Problem Statement	1393
View Verification Case 26 Model	1394
Verification Case 27 Problem Statement	1397
View Verification Case 27 Model	1398
Verification Case 28 Problem Statement	1400

View Verification Case 28 Model	1401
Verification Case 29 Problem Statement	1403
View Verification Case 29 Model	1404
Verification Case 30 Problem Statement	1406
View Verification Case 30 Model	1407
Verification Case 31 Problem Statement	1409
View Verification Case 31 Model	1410
Verification Case 32 Problem Statement	1412
View Verification Case 32 Model	1413
Verification Case 33 Problem Statement	1415
View Verification Case 33 Model	1416
Verification Case 34 Problem Statement	1418
View Verification Case 34 Model	1419
Verification Case 35 Problem Statement	1421
View Verification Case 35 Model	1422
Verification Case 36 Problem Statement	1424
View Verification Case 36 Model	1425
Verification Case 37 Problem Statement	1427
View Verification Case 37 Model	1428
Verification Case 38 Problem Statement	1430
View Verification Case 38 Model	1431
Verification Case 39 Problem Statement	1433
View Verification Case 39 Model	1435
Verification Case 40 Problem Statement	1437
View Verification Case 40 Model	1439
Verification Case 41 Problem Statement	1441

View Verification Case 41 Model	1446
Verification Case 42 Problem Statement	1448
View Verification Case 42 Model	1449
Verification Case 43 Problem Statement	1451
View Verification Case 43 Model	1453
Verification Case 44 Problem Statement	1455
View Verification Case 44 Model	1457
Verification Case 45 Problem Statement	1459
View Verification Case 45 Model	1461
Verification Case 46 Problem Statement	1463
View Verification Case 46 Model	1466
Verification Case 47 Problem Statement	1469
View Verification Case 47 Model	1471
Verification Case 48 Problem Statement	1473
View Verification Case 48 Model	1476
Verification Case 49 Problem Statement	1478
View Verification Case 49 Model	1480
Verification Case 50 Problem Statement	1482
View Verification Case 50 Model	1484
Verification Case 51 Problem Statement	1486
View Verification Case 51 Model	1488
Verification Case 52 Problem Statement	1490
View Verification Case 52 Model	1492
Verification Case 53 Problem Statement	1494
View Verification Case 53 Model	1497
Verification Case 54 Problem Statement	1499

View Verification Case 54 Model	1503
Verification Case 55 Problem Statement	1505
View Verification Case 55 Model	1507
Verification Case 56 Problem Statement	1509
View Verification Case 56 Model	1510
Verification Case 57 Problem Statement	1512
View Verification Case 57 Model	1518
Verification Case 58 Problem Statement	1520
View Verification Case 58 Model	1526
Verification Case 59 Problem Statement	1529
View Verification Case 59 Model	1531
Verification Case 60 Problem Statement	1534
View Verification Case 60 Model	1536
Verification Case 61 Problem Statement	1539
View Verification Case 61 Model	1542
Verification Case 62 Problem Statement	1546
View Verification Case 62 Model	1549
Verification Case 63 Problem Statement	1552
View Verification Case 63 Model	1554
Verification Case 64 Problem Statement	1557
View Verification Case 64 Model	1559
Verification Case 65 Problem Statement	1562
View Verification Case 65 Model	1564
Verification Case 66 Problem Statement	1567
View Verification Case 66 Model	1569
Verification Case 67 Problem Statement	1573

View Verification Case 67 Model	1577
Verification Case 68 Problem Statement	1582
View Verification Case 68 Model	1587
Verification Case 69 Problem Statement	1593
View Verification Case 69 Model	1597
Verification Case 70 Problem Statement	1599
View Verification Case 70 Model	1602
Verification Case 71 Problem Statement	1604
View Verification Case 71 Model	1606
Verification Case 72 Problem Statement	1608
View Verification Case 72 Model	1609
Verification Case 73 Problem Statement	1611
View Verification Case 73 Model	1613
Verification Case 74 Problem Statement	1615
View Verification Case 74 Model	1618
Verification Case 75 Problem Statement	1624
View Verification Case 75 Model	1633
Verification Case 76 Problem Statement	1635
View Verification Case 76 Model	1637
Verification Case 77 Problem Statement	1639
View Verification Case 77 Model	1642
Verification Case 78 Problem Statement	1644
View Verification Case 78 Model	1647
Verification Case 79 Problem Statement	1649
View Verification Case 79 Model	1652
Verification Case 80 Problem Statement	1654

View Verification Case 80 Model	1656
Index	1657

Overview of AFT Fathom

Welcome to AFT Fathom™ - a visual platform for modeling incompressible flow in pipe networks! [See what's new in this version.](#)

AFT Fathom brings increased productivity to the pipe flow engineer by combining traditional engineering hydraulic analysis with an intuitive graphical user interface. By implementing a drag-and-drop method of pipe flow modeling, AFT Fathom simplifies the pipe network modeling process.

AFT Fathom can show both input data and analysis results in visual form, allowing rapid analysis of a model's validity. Identifying poor assumptions, catching typographical or modeling errors, and re-running models are all accelerated because of AFT Fathom's graphical environment.

Whether your pipe system model will be used to evaluate and improve an existing system or to design a new one, AFT Fathom increases your productivity in the modeling process.

What Can AFT Fathom Do?

AFT Fathom can be used to model a wide variety of incompressible flow systems, including:

- Open and closed (recirculating) systems
- Network systems that branch or loop, with no limit on the number of loops
- Pressure fed systems and gravity fed systems
- Pumped systems, including multiple pumps in parallel or in series
- Pumps with variable speed, controlled pressure, controlled flow, and viscosity corrections
- Systems with pressure and/or flow control valves
- Systems with valves closed and pumps turned off
- Heat Transfer and system energy balance
- Systems with variable density and viscosity
- Multiple design cases in a single model file (Scenario Manager)
- Non-Newtonian fluid behavior
- Cost calculation of pump energy usage

AFT Fathom provides many standard loss models for pipe system components, but also allows you to enter your own loss data. Variable loss models that depend on flow are supported.

AFT Fathom's powerful solution engine is based on standard techniques used for many years in industry. The Newton-Raphson method is used to solve the fundamental equations of pipe flow that govern mass and momentum balance. Solutions are obtained by iteration, and matrix methods optimized for speed are employed to obtain convergence.

Where to Start

The best place to start learning about AFT Fathom is to work through our example problems and video tutorials found in the [Fathom Learning Resources](#).

Add-on Module Capabilities

AFT Fathom has several Add-on Modules that extend modeling capabilities. The modules can be used individually or together.

- The [Extended Time Simulation \(XTS\) module](#) allows the engineer to model time varying system behavior.
 - Pump starting and stopping over time
 - Valve opening and closing over time
 - Tank liquid level changes automatically calculated
- The [Goal Seek & Control \(GSC\) module](#) allows the engineer to perform multi-variable goal seeking and simulate control system functions.
- The [Settling Slurry \(SSL\) module](#) allows the engineer to perform the complex property and system interaction calculations associated with settling slurry flows.
 - Modeling of homogeneous and pseudo-homogeneous slurry mixtures
 - Slurry pump performance de-rating
- The [Automated Network Sizing \(ANS\) module](#) automates the process of sizing pipes and ducts in a complex network based on required conditions, such as maximum pressure or minimum flow, to reduce cost or weight.

Who Can Use AFT Fathom

AFT Fathom assumes that the user possesses a good general knowledge of engineering pipe system hydraulics. Even the most advanced and easy-to-use software package cannot make up for a lack of fundamental knowledge on the part of the user. The level of knowledge assumed by AFT Fathom is consistent with that obtained in a typical engineering undergraduate course in fluid mechanics.

Related Topics

FAQ - Frequently Asked Questions.....	56
Engineering Assumptions.....	28
Fathom Learning Resources.....	63
Startup Window.....	64
New in Version 13.....	29
New in Version 12.....	40
New in Version 11.....	43
New in Version 10.....	48
New in Version 9.....	54

Engineering Assumptions

AFT Fathom is based on the following fundamental fluid mechanics assumptions:

- Incompressible flow
- Steady-state conditions (the [XTS Module](#) allows transient modeling through steady-state snapshots)
- One-dimensional flow
- System is fluid-full
- Fluid properties are single phase
- No chemical reactions

Related Topics

Overview of AFT Fathom.....	27
FAQ - Frequently Asked Questions.....	56
Fathom Learning Resources.....	63
Startup Window.....	64
New in Version 13.....	29
New in Version 12.....	40
New in Version 11.....	43
New in Version 10.....	48
New in Version 9.....	54

New in Version 13

View the links below for key updates from past versions:

[New in Version 12](#), [New in Version 11](#), [New in Version 10](#), [New in Version 9](#)

Workspace Layers

The [Workspace Layers](#) feature provides a powerful new tool for customizing and viewing model appearance, input data, and results - all in one place. The Workspace Layers toolset significantly enhances the model building process, providing unparalleled customization options. You can easily update pipe/junction size, color, display text, visibility, and more from a centralized location on the Quick Access Panel. Workspace Layers surpass the capabilities of the previous Visual Report feature. With Layer Settings, you now have easy control over Color Mapping and the visibility of both Input and Output parameters. Any change you make to these settings will have an immediate impact on the model's display, providing real-time feedback. The dynamic combination of unique Layers offers an incredibly flexible framework, allowing you to display precisely what you desire on the Workspace. You can easily toggle the visibility of individual layers on or off, and the state of all layers can be saved to a Layer Preset for future use.

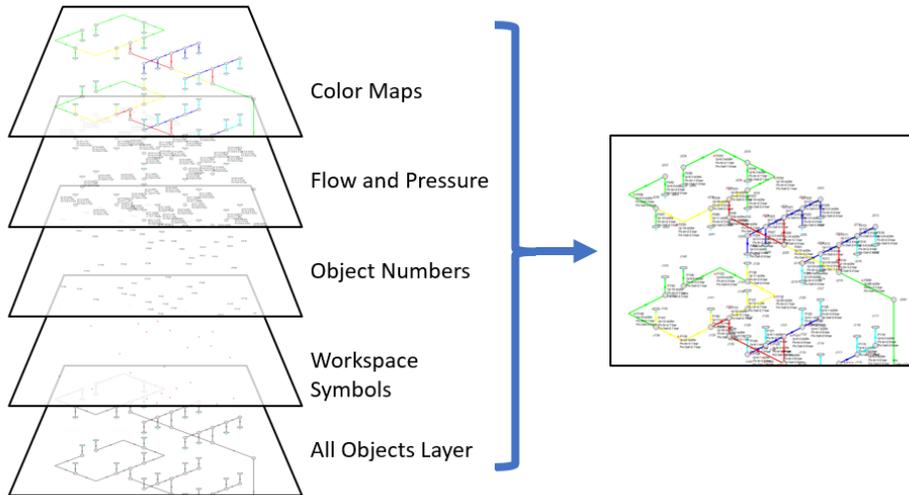


Figure 1: Workspace Layers can display input and output values superimposed over the model, show Color Maps, and control visibility of objects

4-Component Slurry Model

A new [4-Component Slurry Model](#) for the SSL module, developed by leading industry experts, captures complex slurry regimes more accurately. The 4-Component Model introduces a division of slurry solids into four distinct groups, or components, based on the particle size distribution. Each component corresponds to a specific slurry behavior; Carrier Fluid, Pseudo-homogeneous, Heterogeneous, or Fully Stratified. Unlike simpler models that account for only one or two slurry regimes, the 4-Component Model greatly enhances modeling flexibility by automatically determining the effects contributed by each component regime. The model was developed by industry-leading experts and is respected for its accuracy in part because the individual components are based on well established sub models. The 4-Component Model offers a comprehensive and advanced approach to slurry modeling, enabling more accurate and detailed analysis of slurry behavior in various particle size distributions.

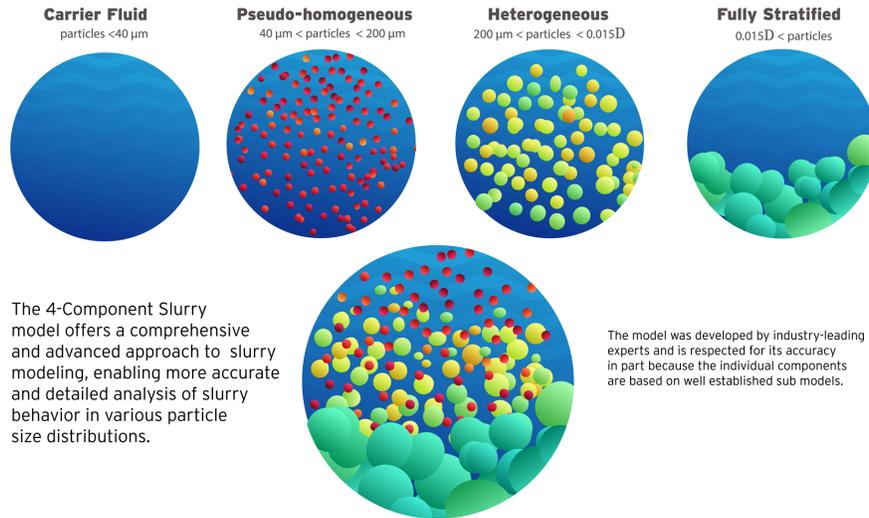


Figure 2: The 4-Component Slurry Model considers combined effects from four slurry regimes

Improved Valve Properties Window

The new [Valve Properties window](#) greatly enhances clarity and usability when it comes to modeling valves. The window now clearly distinguishes between different data sources such as User Specified, Characteristic, or Handbook, making it easier to understand the available options. The form has been streamlined to focus on the most essential data, providing a clear understanding of what is required to define a valve accurately. Valve Characteristics, represented by an Open Percent vs. Flow Coefficient table, have been brought front and center. While using a Characteristic is optional, it is highly recommended as it ensures that the valve behavior aligns with real-world scenarios, particularly for valve transients. The process of selecting a pre-defined Characteristic Source has been simplified, which helps obtain more accurate values in case manufacturer data is unavailable.

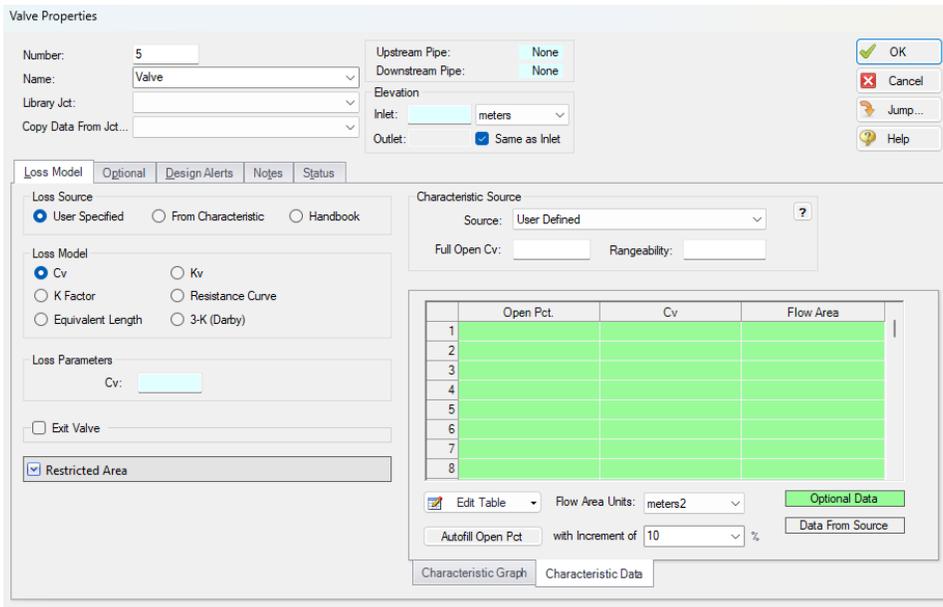


Figure 3: Redesigned Valve Properties window

Streamlined Design Alerts

The [Design Alert Manager](#) has been reorganized to improve the process of creating and applying user-defined alerts. Now, you can effortlessly create or edit a Design Alert with just a few clicks. As you modify settings, you will immediately see the criteria you are applying, ensuring a transparent experience. We have also reduced the amount of text on the form, while ensuring grammatical clarity. These improvements aim to make it easier to understand the limits being placed on the design.

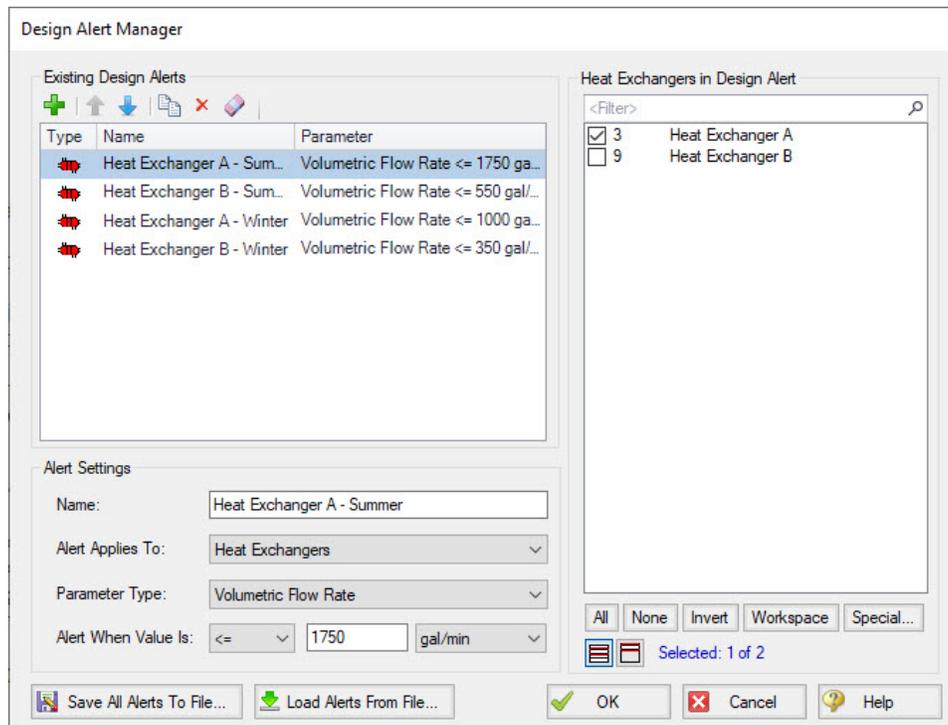


Figure 4: Steamlined Design Alerts Manager

Other Notable Features

- **Custom Junction Icons** - Use your own images for [junction icons](#). Utilize any image as a junction icon, simply by right-clicking on the Toolbox icon for the junction, selecting Add Image, and browsing to the image file. This functionality enables you to use standard P&ID icons or custom images to visually represent your equipment. By using recognizable icons, you can improve the clarity and comprehension of model results, especially for those who may not be familiar with the default AFT junction icons.

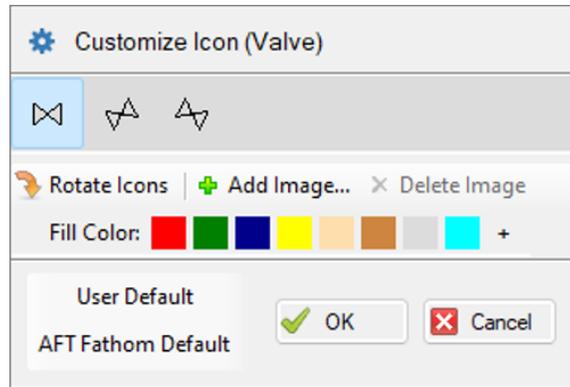


Figure 5: The Customize Icon window allows a custom image to be applied to any junction

- **Isometric Freeform Drawing Mode** - Draw on the isometric grid without forcing any particular pipe routing.

The Isometric drawing tools have been enhanced by adding a new Isometric Freeform [drawing mode](#). This new drawing mode allows you to snap junction and pipe anchors to the Isometric grid, while having the flexibility to draw pipes that do not necessarily align with the isometric gridlines. This allows you to easily draw complex systems where not all piping follows a Cartesian direction.

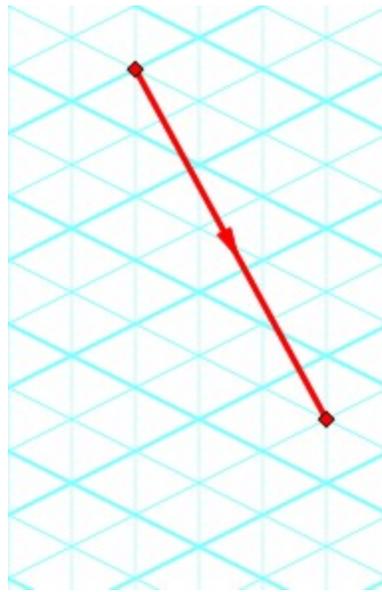


Figure 6: Isometric Freeform drawing mode does not force pipes to follow gridlines

- **New Annotations** - There are new shapes and a line tool, along with a more advanced [Annotation](#) editing panel.

New Annotation shapes and a line tool have been introduced to allow more flexible markup such as revision numbers or change requests. Editing of Annotations has been revamped with the introduction of a live-edit Quick Access Panel, which provides immediate visual feedback of the changes you make, and simple global editing of all selected Annotations. The new edit panel also

offers expanded functionality, providing you with more customization than was possible in the past.

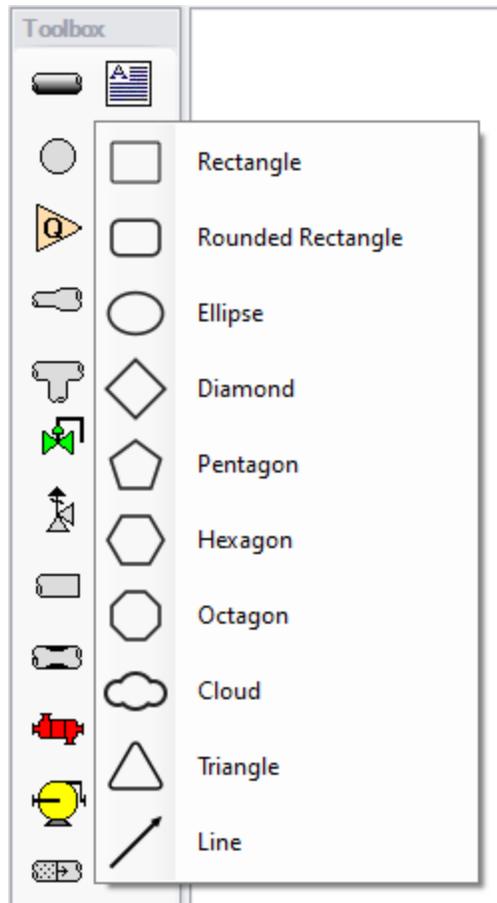


Figure 7: New Annotation options

- **Set Workspace Print Area** - Show [printing](#) page borders on the Workspace and customize the printing area.

The new Printer Page Layout tool offers a convenient and effortless way to adjust the printing area according to your preferences. With a clear and intuitive visual representation, you can easily identify where page breaks will occur. Now, you can confidently print a specific section or the entire model at your desired scale, eliminating the need for trial and error.

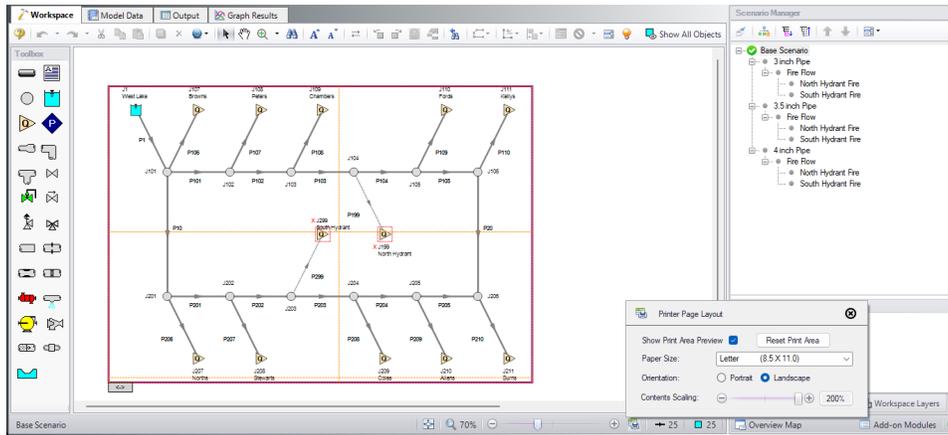


Figure 8: The Printer Page Layout feature allows the user to set the Workspace print area

- **Customize Parameters Available to Graph from the Workspace** - New [User Options](#) allow custom selection of any graph parameter on the right-click Workspace context menu.

Did you know you can easily graph results from the Workspace by right-clicking on a pipe or junction? This functionality has existed for a long time, but it was previously limited to a predefined set of parameters. Now, you can specify what graph types you would like to see in these menus to simplify your analysis process.

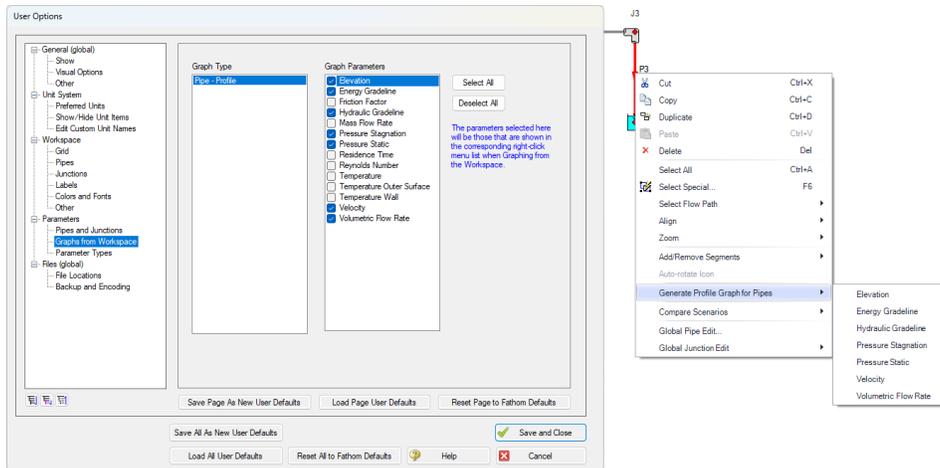


Figure 9: The right-click context menu for pipes on the Workspace can have the graphing parameter options modified

Miscellaneous Improvements

- **Modeling Additions and Improvements**
 - **Laminar Corrections** - Laminar corrections are now off by default, messages indicate if they were applied or are recommend, and output parameters indicate the correction strength.
 - **Prevent Backflow at Sprays** - New "No Inflow" Special Condition for Spray junctions.
 - **Zero-Length Connectors** - Zero-Length Connectors now behave like standard pipes for most features, including selection and graphing tools.
 - **Distinguish User and AFT Fluids** - New category for user-defined fluids, making it easier to find custom entries.
- **Importing of PCF, EPANET, GIS, and CAESAR II Models**
 - **Bend r/D** - Import PCF bends with appropriate r/D values.
 - **Speed Increase** - Importing is up to 10x faster in some cases.
 - **Defaults and Display** - Better defaults and display options for imported models.
 - **Pipe Materials** - Improved ability to link AFT pipe materials to imported pipes.
 - **CAESAR II Node Names** - Display imported CAESAR II node names as junction names.
- **Behind the Scenes Improvements**
 - **Graphing Optimization** - Significant improvements to backend graphing code which provides additional stability now and more capabilities in the future.
 - **eLicense Access Improvements** - Added a "heartbeat" functionality to eLicenses which improves security and the release of license seats not in use.
 - **Output** - Significant improvements to backend output code which provides additional stability now and more capabilities in the future.
 - **Timed Recovery File** - Automatically generate a backup of your *unsaved* model that can be used to recover unsaved work in the event of unexpected application closure. This is separate from the *.bak files which are only generated *after* saving a model.
- **New and Improved Output Parameters and Messages**
 - **Pump RPM** - Report pump speed in RPM, in addition to percent.
 - **Dimensionless Heat Transfer Parameters** - Grashof, Nusselt, Prandtl, and Rayleigh Numbers have been added as Output parameters.
 - **Deep Laminar Caution** - Simple loss models do not account for exceptionally low laminar flow, this [new message provides a caution](#) if you enter this range.
 - **POR/AOR in Pump Summary** - Show the values for Preferred and Allowed Operating Ranges in the Pump Summary.
 - **Event Messages in New Format** - Events are now shown in a cleaner, collapsible format similar to Warnings.
 - **XTS Module Tank Overflow and Drain Warnings** - New messages to clearly indicate if a Tank has [overflowed](#) or [drained](#) during the transient.

Related Topics

Overview of AFT Fathom.....	27
FAQ - Frequently Asked Questions.....	56
Engineering Assumptions.....	28
Fathom Learning Resources.....	63
Startup Window.....	64
New in Version 12.....	40
New in Version 11.....	43
New in Version 10.....	48
New in Version 9.....	54

Interface Improvements for Fathom 12

There were many improvements made to the interface in Fathom 12. These changes primarily involve consolidating inputs with the new Analysis Setup window, and updating the functionality of databases, which have been renamed as libraries.

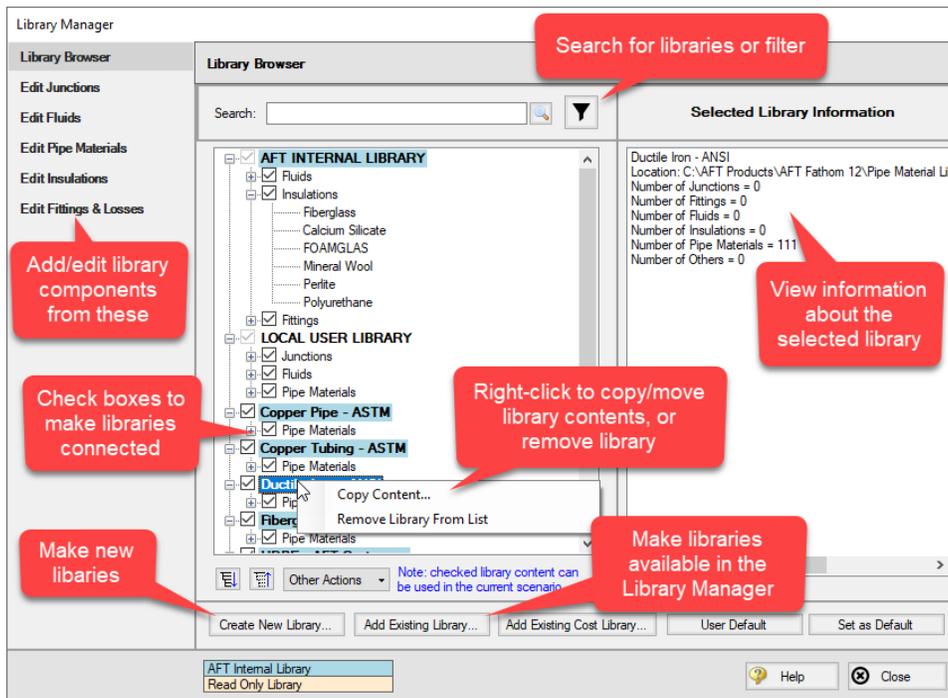
Analysis Setup Sources in Fathom 11

- **System Properties:** The System Properties window has been split up into the [Fluid Properties group](#) and the [Environmental Properties group](#). The System Properties icon is still available from the Analysis menu and toolbar to directly open the Fluid Properties panel.
- **Solution Control:** The Solution Control parameters have primarily moved to the [Steady Solution Control group](#). The Special Controls have moved to the [Advanced Parameters panel](#) under the Miscellaneous group
- **Checklist:** The checklist which was shown in the Quick Access Panel has been replaced by the status icons in the [Analysis Setup window](#). Red exclamation marks denote which inputs are incomplete
- **List Undefined Objects:** The [undefined objects list](#) has been moved to the [Pipes and Junctions group](#). The undefined objects list can also still be viewed next to the Workspace by opening it from the View menu.
- **Cost Settings:** Cost calculations were previously enabled from the Analysis menu and defined in the Cost Settings window. Enabling and defining cost settings has now been moved to the [Cost Settings group](#)
- **NFPA Configuration:** Enabling NFPA reports has moved from User Options to the [NFPA panel](#) under the Miscellaneous group. In addition selecting an NFPA group has moved from the Tools menu to the NFPA panel.
- **Extended Options:** Enabling loss model options such as equivalent length that was previously in the Extended Options window has been moved to the **Junction Loss Models panel** under the Miscellaneous group
- **Add-on Modules Activation:** The activation window for add-on modules has moved to the [Modules panel](#)
- **Goal Seek & Control Manager (GSC Module Only):** Settings have been moved to the [Goal Seek and Control group](#)

- **Transient Control (XTS Module Only):** Settings have been moved to the [Transient Control group](#)
- **Slurry Data (SSL Module Only):** Settings have been moved to the [Fluid Properties group](#)
- **Sizing Summary (ANS Module Only):** The Sizing Summary has been added to the [Sizing Summary panel](#) in Analysis Setup. The Sizing Summary can also still be viewed from the Sizing window. No other ANS module interface changes have been made.

Database Improvements

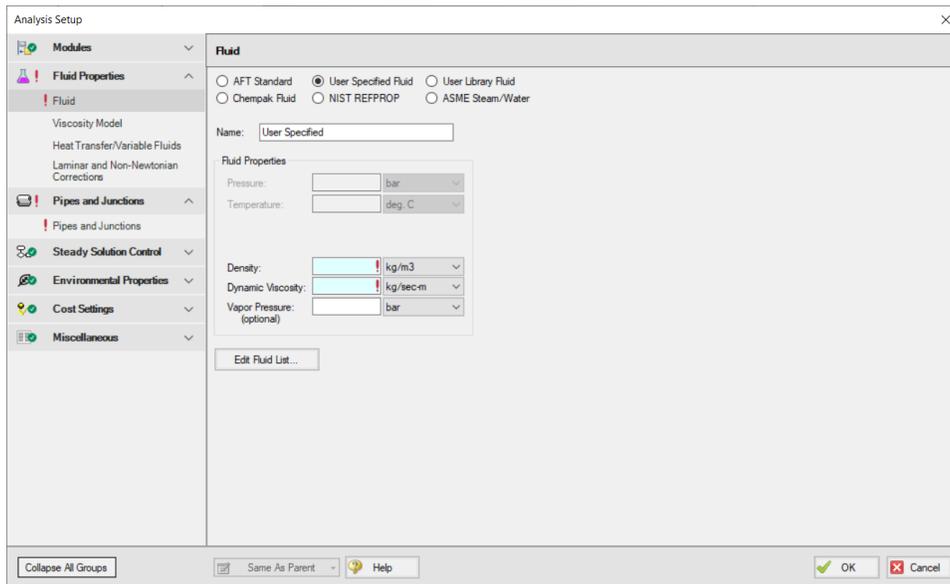
- Databases have been renamed as libraries throughout the interface to better reflect their functionality
- Connecting, creating, and transferring content between libraries can all be done from the Library Browser, rather than switching between tabs in the Database Manager to do this
- Viewing the contents of libraries can also be done directly in the Library Browser, rather than opening a separate Browse Database Content window
- Windows to edit and add components to libraries can all be accessed directly from the Library Manager rather than having to navigate solely through the Library menu



New in Version 12

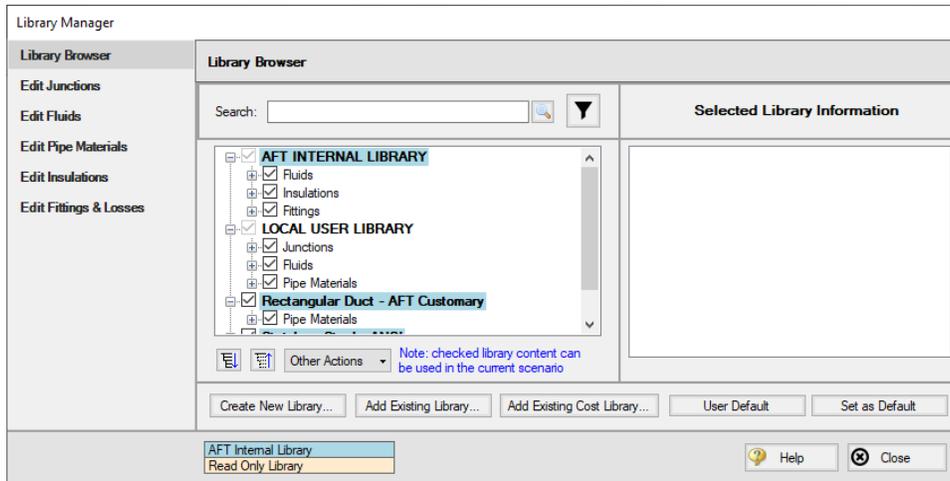
Analysis Setup

[Analysis Setup window](#) consolidates windows for System Properties, Solution Control, Activate Modules, Extended Options, Goal Seek & Control Manager, and Transient Control all into one location. This allows users to activate and enable modules, specify fluids, select a viscosity model, enable heat transfer, change solution tolerances or iterations, and so on, all from a single window. For a list of windows that have merged into the Analysis Setup window see the topic on [Interface Improvements for Fathom 12](#).



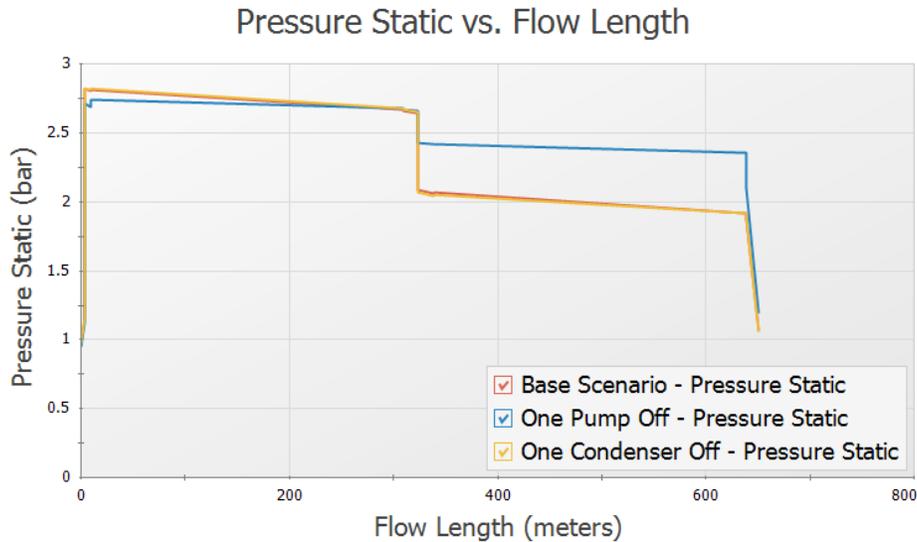
Libraries

The database feature interface was completely overhauled to improve workability. The new system is called the [Library Manager](#) and significantly improves the ease and workability for users to browse and connect input information, now called libraries, into their model. For a list of differences between the Database and Libraries interfaces see [Interface Improvements for Fathom 12](#).



Multi-Scenario Graphing

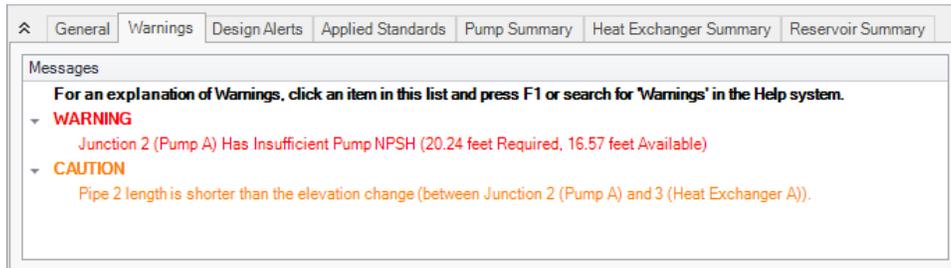
Profile graphs (and Transient Graphs in the XTS module) can now include output from multiple scenarios, making it easy to compare the results along a flow path for different scenarios. See the Creating Multi-scenario Graphs tutorial.



Improved Organization of the Warnings in the Output

An improved display of the warnings and design alerts makes it easier to review and understand information in the following ways:

- Critical Warnings, Warnings and Cautions are color-coded and grouped by type in collapsible sections
- [Design Alerts](#) are now shown on a separate tab in the General Section
- Double-clicking messages will take user to the related pipe/junction in the Workspace
- Warnings, Cautions and Design Alerts are now displayed for all scenarios when using the Multi-scenario Output feature



Miscellaneous Improvements

- **Pipes/Junctions**
 - Pipes can be defined as zero-length connectors to connect two junctions which are directly flanged together
 - [Submerged pump](#) can now be defined using surface pressure and pump depth
 - New [Pump As Turbine \(PAT\)](#) option on the Pump Properties window
 - Valve transients in the XTS module can be defined as open percent vs time
- **Workspace and Interface**
 - Pipes and Junctions can be set the Same as Parent directly from the Workspace
 - Custom Unit names can be defined in the [Unit System](#) section of the User Options window
 - Pipe/Junction notes can be searched in [Select Special](#)
 - Special Conditions can be directly changed from the toolbar for junctions that have multiple special condition options
- **Output and Reporting**
 - Show the number of warnings and design alerts in the [batch run](#) window
 - Heat Transfer output parameters added to Visual Report
- **Miscellaneous**
 - [Herschel-Bulkley non-Newtonian viscosity model](#) added for representing Power Law fluids which have an apparent yield stress
 - List Junctions that have Special Conditions change in Solution Progress Window
 - [Silent Batch runs](#) can be enabled to prevent notifications from appearing over other active windows

Related Topics

Overview of AFT Fathom.....	27
FAQ - Frequently Asked Questions.....	56
Engineering Assumptions.....	28
Fathom Learning Resources.....	63
Startup Window.....	64
New in Version 13.....	29
New in Version 11.....	43
New in Version 10.....	48
New in Version 9.....	54

New in Version 11

Scenario Comparison Tool

- Quickly get a comparison of different scenarios within a model file
- Useful to double-check input results for old models, or for troubleshooting purposes
- For a detailed description, see [Scenario Comparison Tool Overview](#)

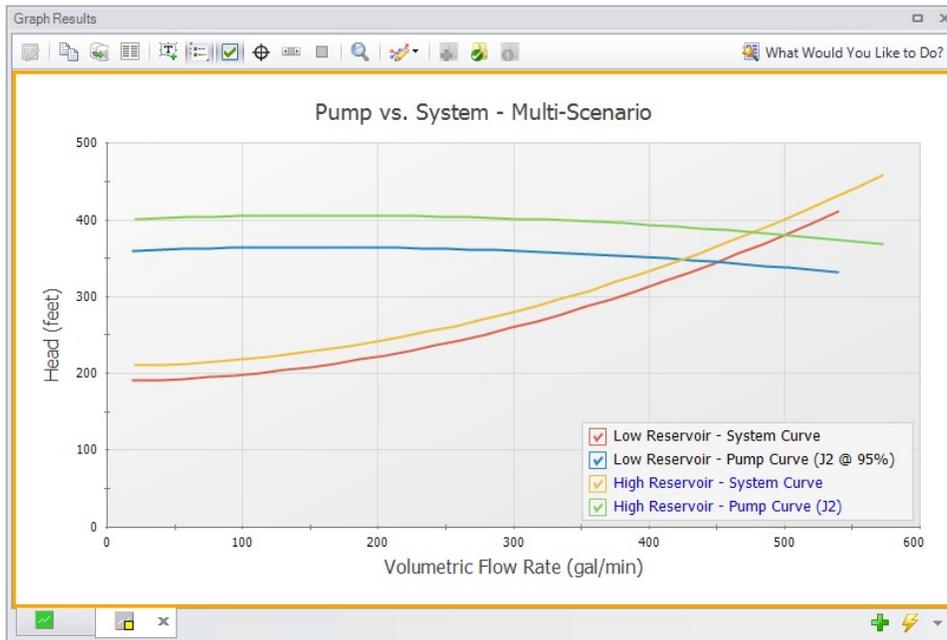
Item	Parameter	Scenario		
		South Hydrant Fire	3 inch Pipe	North Hydrant Fire
J109 (Chambers)	Flow	2 gal/min	10 gal/min	2 gal/min
J110 (Fords)	Flow	2 gal/min	10 gal/min	2 gal/min
J111 (Kellys)	Flow	2 gal/min	10 gal/min	2 gal/min
J199 (North Hydrant)	Special Condition	Flow Off	Flow Off	None
J201 (Branch)		No Differences		
J202 (Branch)		No Differences		
J203 (Branch)		No Differences		
J204 (Branch)		No Differences		
J205 (Branch)		No Differences		
J206 (Branch)		No Differences		
J207 (Norths)	Flow	2 gal/min	10 gal/min	2 gal/min
J208 (Stewarts)	Flow	2 gal/min	10 gal/min	2 gal/min
J209 (Coles)	Flow	2 gal/min	10 gal/min	2 gal/min
J210 (Allens)	Flow	2 gal/min	10 gal/min	2 gal/min
J211 (Burns)	Flow	2 gal/min	10 gal/min	2 gal/min
J299 (South Hydrant)	Special Condition	None	Flow Off	Flow Off
Solution Control (3 of 3 compared)				
	Tolerance	No Differences		
	General	No Differences		

Only parameters with differences will be shown. If all chosen parameters for an item are the same, then the item will say 'No Differences'. Parameters containing tabular data (e.g. polynomial constants) can be viewed by right-clicking the parameter's row.

Item	Parameter	Highlight Unique Values
All Parameters Same Value	Same Value Across Scenarios	No <input type="checkbox"/> Yes <input checked="" type="checkbox"/>
At Least 1 Parameter Difference	Different Values Across Scenarios	

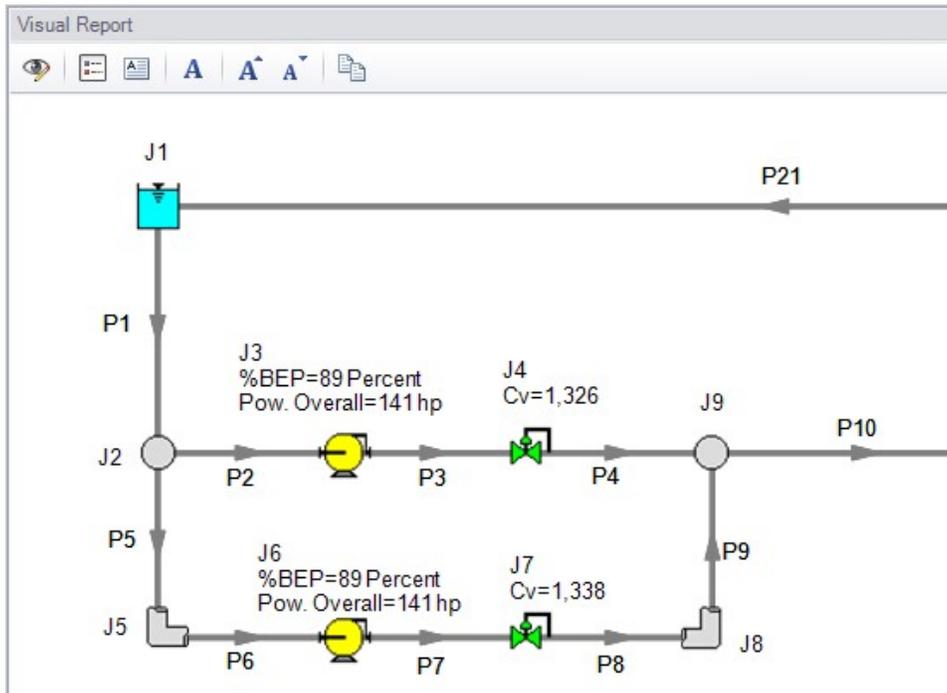
Multi-scenario Pump vs. System Curves

- Pump vs System curves from different scenarios can be plotted on the same graph within AFT Fathom
- Easily compare changes to pump operation in different cases
- For more information see [Multi-Scenario Pump vs. System Curves Tutorial](#)



Junction- specific Parameters in Visual Report

- Output such as the percent of BEP for a pump or the Cv of a control valve can now be displayed in the Visual Report



Notes in Output

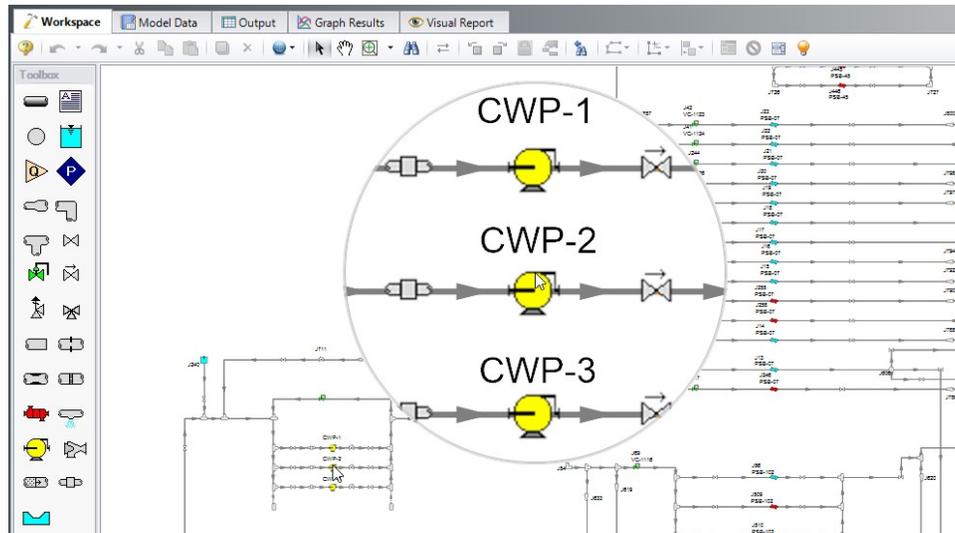
- Notes can be added to any cells in the [Output](#) to save observations and comments directly with the model file

Pipe	Vol. Flow Rate (gal/min)	Velocity (feet/sec)	P Static Max (psig)	P Static Min (psig)	Elevation Inlet (feet)	Elevation Outlet (feet)	dP Stag. Total (psid)	dP Static Total (psid)	dP Gravity (psid)	P Static In (psig)	P Static Out (psig)	
100	8,031.9	8.854	1.893	-0.5250	6.000	0.00	-2.4177	-2.4177	-2.585	-0.5250	1.893	
101	8,031.9	8.854	26.446	26.2362					2094	26.4456	26.2362	
102	16,064.8	7.803	26.353	24.0517	Minimum pressure matches plant measurements from calibration data within 5%]				3017	0.000	26.3535	24.0517
103	830.6	5.140	24.283	24.1012					1813	0.000	24.2826	24.1017
106	834.3	5.162	15.477	15.2978					1794	0.000	15.4772	15.2978
107	15,233.8	8.536	23.971	23.6073					3642	0.000	23.9715	23.6073
108	15,301.5	8.574	15.354	14.9853					3685	0.000	15.3538	14.9853
109	16,136.1	7.838	15.066	12.7786					2873	0.000	15.0659	12.7786
110	10,628.3	8.032	12.758	12.6495					1086	0.000	12.7580	12.6495
111	5,244.3	7.095	12.744	12.6232					1207	0.000	12.7439	12.6232
112	5,244.3	11.074	12.000	11.5731					1200	0.000	12.0000	11.5731

Heat Exchanger	Name	P Static In (psig)	P Static Out (psig)	P Stag. In (psig)	P Stag. Out (psig)	Vol. Flow Rate Thru Jct (gal/min)	Mass Flow Rate Thru Jct (lbm/sec)	Loss Factor (K)
105	HTC #110 CL	24.1012	15.40	24.278	15.05	830.6	114.0	48.750

New Workspace Functionality

- Multi-level Undo** - Drawing, arranging and moving objects in the Workspace now allows for multiple levels of undo
- Workspace Magnifier** - The [magnifier tool](#) provides an enlarged view of sections of the model to make it easier to manage and navigate large models



- **Automatic Junction Rotation** - Fathom will automatically rotate junctions to visually align with neighboring pipes to speed-up model formatting
- **Combine Pipes** - a series of pipes and junctions can be [combined](#) to simplify a model, with any losses being imported to the Pipe Fittings and Losses tab, and any elevations being imported as Intermediate Pipe Elevations
- **Split Pipes by Intermediate Elevations** - Pipes with intermediate elevations can be split with branches at intermediate elevation points (see [Joining and Splitting Pipes](#))
- **Select Shortest Flow Path** - The shortest path between two selected junctions can be found using this feature in the [Edit menu](#)

Diagnostics Window

- Provides a list of features and settings which might cause issues as a starting point for model troubleshooting
- See the [Diagnostics Window](#) topic for more information

More Improvements

- **Overall Performance**
 - Improved intelligence on when a model needs to be saved and re-run to preserve the output
 - Faster model loading speed
 - Improved language refresh speed
 - New and updated themes
- **Workspace**

- The Undefined Objects panel remains open when pipes/junctions are being edited
- Double-clicking a junction allows more than one of that junction to be placed in the Workspace at a time
- Snapping can be turned off for pipes when the isometric [drawing mode](#) is enabled
- **Output**
 - [Output](#) parameters can now be filtered/searched for simpler selection
 - The average thermal conductivity used for calculations can be displayed in the Output
- **Visual Report**
 - Visual Report Control has been redesigned to better filter parameters
- **Pipes**
 - [Heat tracing](#) now allows the entry of power/length and number of turns to allow Fathom to calculate the heat flux directly
- **Junctions**
 - Updated [Tee/Wye Properties window](#) which clarifies the input and automatically defines the pipe arrangement according to the direction and orientation on the Workspace
 - Built-in K values are now included for [bends](#) with r/D between 0.5 and 1
 - Notes can be globally edited for junctions
- **Design Alerts**
 - [Design Alerts](#) can be [saved to and loaded from a file](#) to quickly apply common Design Alerts to multiple models
- Ability to [import custom Fittings & Losses](#) from a file

Related Topics

Overview of AFT Fathom.....	27
FAQ - Frequently Asked Questions.....	56
Engineering Assumptions.....	28
Fathom Learning Resources.....	63
Startup Window.....	64
New in Version 13.....	29
New in Version 12.....	40
New in Version 10.....	48
New in Version 9.....	54

New in Version 10

New & Improved Microsoft Excel Integration

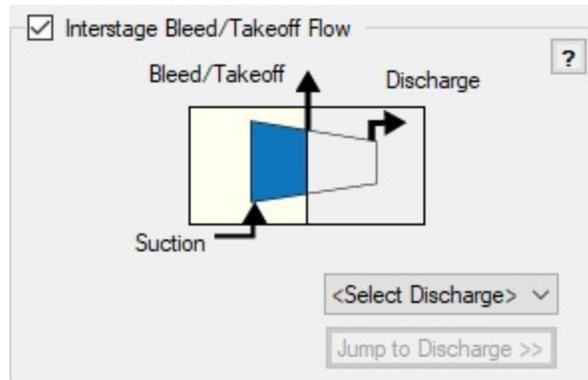
- Improved keyword based method for importing model changes from an Excel Worksheet
 - Use the included template Worksheet to quickly and easily define model changes
 - Change **multiple scenarios at once**
 - For a detailed description see [Importing From Excel](#)

	A	B	C	D	E	F	G	H
	Comment	Apply	Object Type	Object Number	Parameter	Change Code	Value	Scenario Path Name
2	HEX heat rate set	Yes	Heat Exchanger	67	Heat Rate	Set equal to value	4000	High Heat
3	HEX heat rate set	Yes	Heat Exchanger	66	Heat Rate	Set equal to value	3200	Low Heat
4	HEX heat rate set	Yes	Heat Exchanger	65	Heat Rate	Set equal to value	3700	Normal Heat
5	FCV setpoint set	Yes	Control Valve	71	Control Setting	Set equal to value	300	
6	FCV setpoint reduce by 20 gpm	Yes	Control Valve	72	Control Setting	Change by value (+/-)	-20	
7	FCV setpoint increase by 10%	Yes	Control Valve	73	Control Setting	Change by percent (+/-)	10	
8	Pump Speed set	Yes	Pump	59	Pump Speed (%)	Set equal to value	90	

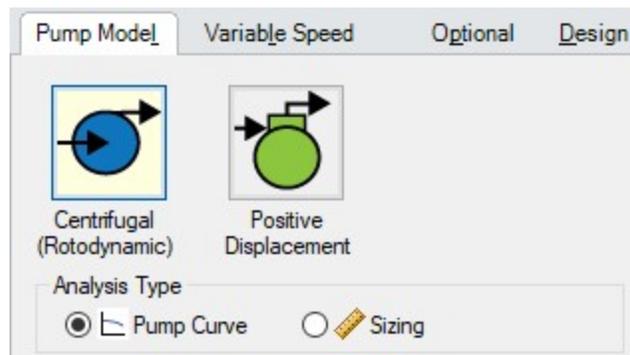
- New feature for directly exporting specified Output data to new or existing Excel Workbooks.
 - Export only the data that is desired - no need to sort through unnecessary data
 - Specify exactly where the data will end up - insert data into a pre-formatted Workbook
 - Automatically export after a model run
 - Export data from multiple scenarios with a [batch run](#)
 - For a detailed description see [Exporting To Excel](#)

New Isometric Drawing Mode

- New [Isometric Pipe Drawing Mode](#)
 - Show and Snap to an isometric grid on the Workspace
 - View models in an intuitive 3D layout, and easily build from existing isometric drawings



- New [Positive Displacement Pump](#) Model
 - Clarifies the specification of Positive Displacement Pumps
 - Include Acceleration Head Loss calculations
- Added Pump Curve and Sizing Analysis Types to Centrifugal Pump Model
 - Clarifies required data and model intention
 - For a detailed description see [Pump Properties Window](#)



- Specify a centrifugal pump's Preferred Operating Region (POR) and Allowable Operating Region (AOR)
 - Quickly determine if a pump's behavior is acceptable
 - Automatically calculate a POR from *ANSI/HI 9.6.3-2017 - Rotodynamic Pumps - Guideline For Operating Regions*
 - For a detailed description see [Pump Configuration > Configuration Data](#)

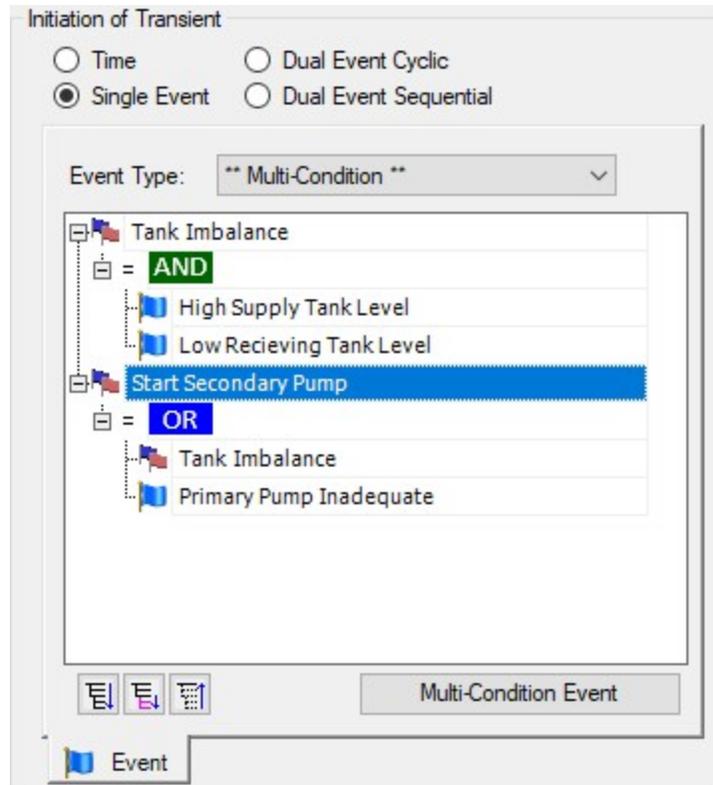
New Thermal Models

- Two new Heat Transfer Models for Pipes
 - [Convective Heat Transfer With Heat Tracing](#) - model the addition of heat between a pipe wall and an insulation layer

- [Buried Pipe](#) - model the thermal effects of burying a pipe
- Calculate a Forced Convection Coefficient from ambient fluid velocity and temperature with the [Churchill-Bernstein correlation](#)

New Multi-Condition Events for XTS Module

- Create complex triggers for transient events in the [eXtended Time Simulation \(XTS\) Module](#)
 - Only start a transient event when multiple conditions are met
 - Utilize logical conditions - AND, OR, and NOT - to capture the effects of complex control systems
 - For a detailed description see [Multi-Condition Events](#)



Improved Startup Window

- Improvements to the capability and usability of the Startup Window
 - Set modeling preferences including - [active modules](#), [working fluid](#), [units](#), [language](#), 2D or Isometric [drawing mode](#), [default pipe material](#), and [NFPA settings](#)
 - Displayed units can now be easily refined by selecting Common Units and Industry-Specific Units

New Integration with Common Piping Layout formats

- Save modeling time by importing existing Piping Layouts from various neutral file types
 - Import [CAESAR II Neutral Files](#) (.cii)
 - Import [Piping Component Files](#) (.pcf)
 - Import and Export [EPANET Files](#) (.inp)
 - These common file formats are generated from many common engineering programs such as AutoCAD Plant 3D, SmartPlant, PDS, CadWorx, and many more.

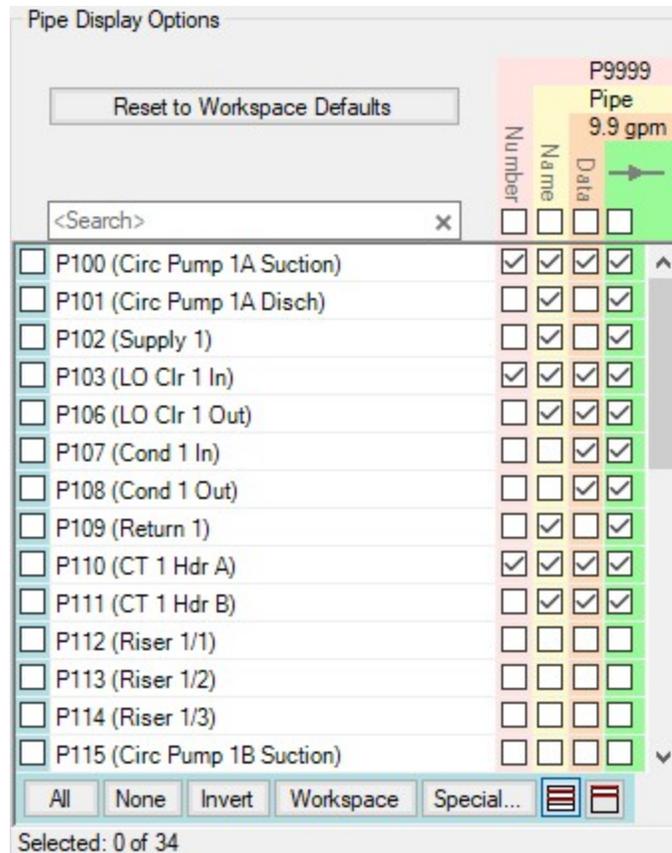
Updated Standards

- Output displays [Applied Standards](#) - standards that were used in calculation of model results
- [Relief Valve loss](#) can be specified with Designation Size per *API 526 - Flanged Steel Pressure Relief Valves*
- Updates to Standards
 - Updated to *HI/ANSI 12.1-12.6-2016 Rotodynamic (Centrifugal) Slurry Pumps for Nomenclature, Definitions, Applications, and Operation* for use in [Slurry De-rating](#)
 - Updated to *HI/ANSI 9.6.7-2015 Rotodynamic Pumps - Guideline for Effects of Liquid Viscosity on Performance* for use in [Viscosity Corrections](#)
 - Updated MIT Equation for Crude Oil to latest version specified in Pipeline Rules of Thumb for use as an additional [pressure loss model](#)

More Improvements

- **General**
 - Many new [default Pipe Materials](#)
 - [Intermediate pipe elevations](#) can now be entered as Length Along Pipe
 - [Control Valve](#) setting for Always Control (Never Fail) can be set to a default to off
 - Increased the loading speed for large models
 - Applying [Design Alerts](#) when using the [XTS module](#) now shows Design Alert Violations in transient Output tables, and all time steps that violate the Design Alert are highlighted
- **Workspace**
 - The [Workspace background color](#) can be changed based on the level of the current scenario - additional levels have been added
 - [Annotation](#) background opacity can be changed
 - New [keyboard shortcuts](#) to zoom (CTRL + +/-) and pan (CTRL + arrow keys) the Workspace
 - [Full Scenario Path Name](#) can be copied to clipboard by right-clicking a scenario

- [Selection box visually indicates](#) the behavior of the selection - select everything intersecting (right to left) or only objects completely inside (left to right)
- Pipes with Fittings & Losses defined now display a [special character](#) next to their number (default of &)
- **Output**
 - [Design Alerts can be created directly from the Output](#) by right-clicking on a result - immediately select the appropriate object and parameter.
- **Model Data**
 - [Quickly locate a specific object on the Workspace](#) by right-clicking the item in Model Data
 - Added additional parameters such as insulation thickness
- **Visual Report**
 - Reworked Display Option controls for Pipes and Junctions in Visual Report Control



Related Topics

Overview of AFT Fathom.....	27
FAQ - Frequently Asked Questions.....	56
Engineering Assumptions.....	28

Fathom Learning Resources.....	63
Startup Window.....	64
New in Version 13.....	29
New in Version 12.....	40
New in Version 11.....	43
New in Version 9.....	54

New in Version 9

General Interface

- Improved tab interface and enhanced Quick Access Panel functionality
- Improved search includes pipe and junction notes, names and numbers
- Output reports in German and Chinese languages
- Files are automatically locked when opened to prevent multiple users from accidentally opening the same file
- Global edit select all items within a section

Graph Results

- Multiple Graphs Sets can be added to Graph Groups for easy generation and display
- Multiple graph tabs on main Graph tab
- Multiple graphs tiled on same Graph Group tab
- Easy formatting of titles, axis labels, and plot lines
- Plot series lines can be turned on/off for quick inspection
- Graph controls now located on the Quick Access Panel for easy graph definition and changes
- Junction location shown on profile plots
- Annotations can be placed on graphs

Scenario Manager

- Scenario manager shows which scenarios have output
- Ability to create child scenarios without results saving duplication and load time

Workspace

- Scenario manager shows which scenarios have output
- Ability to create child scenarios without results saving duplication and load time

Output

- Results Diagrams which graphically show data for pipe heat transfer, pumps and heat exchangers
- Default colors for multi-scenario output.
- Support for NFPA output reporting

Visual Report

- Automatic Color Map generation

Pipes

- Design Alerts are specified and named globally and applied to pipes
- New Design Alerts available

Junctions

- Improved Relief Valve specification
- Improved clarity for data entry in the Reservoir/Tank Property window
- Submerged pumps
- When the speed is set for a pump, both the original and modified pump curves are shown on the Pump Property window
- New Weir junction
- Design Alerts can be applied to junctions

Solver

- 3-K method for laminar flow through fittings
- Equivalent Length option available for pipes, valves and bends

Modules

- GSC
 - Pipes now have GSC variables available such as roughness, design factor, insulation thickness, Reynolds number
 - New and improved searching method for multi-variable goal seeking – useful for model calibration

- Variables and goals can be reordered
- New goals and variables added

Related Topics

Overview of AFT Fathom.....	27
FAQ - Frequently Asked Questions.....	56
Engineering Assumptions.....	28
Fathom Learning Resources.....	63
Startup Window.....	64
New in Version 13.....	29
New in Version 12.....	40
New in Version 11.....	43
New in Version 10.....	48

FAQ - Frequently Asked Questions

How has AFT Fathom Been Verified?

There are several aspects to AFT Fathom verification.

First, AFT Fathom has been compared to a wide variety of published results. Models, comparisons, and explanations for such published predictions are given in the [Verification](#) section of this site and the Verification folder installed with AFT Fathom.

Second, it can be verified that the AFT Fathom predictions agree with the fundamental equations for networks. See the [Network Solution Methodology](#) topic.

Finally, AFT Fathom predictions have been compared to test data and other analytical methods on numerous occasions and good agreement has been shown.

Can AFT Fathom Be Used to Model Gas Systems?

Yes, depending on the gas system (see Walters, 2000). For incompressible gas systems, AFT Fathom is perfectly appropriate. For compressible flow gas systems, a more sophisticated tool such as AFT Arrow should be considered.

Can AFT Fathom Model Two-Phase Flow?

No, AFT Fathom only models single phase flow and assumes all pipes are liquid full.

How Does AFT Fathom Account for Heat Transfer in Pipe Systems?

Heat transfer effects show up in several places in AFT Fathom. First, for pipes users can input pipe and insulation data thermal conductivity data and thickness, as well as external convection coefficients. The thermal conductivity data can be constant or temperature dependent. These heat transfer calculations

and the associated physical properties are determined for each pipe. Second, heat transfer can occur in Heat Exchanger junctions. Here the user assigns a heat load or chooses a heat transfer model for the heat exchanger. AFT Fathom performs the appropriate energy balance calculation across the heat exchanger. Third, heat transfer occurs in pumps due to inefficiencies.

Finally, at each branching location in the network an energy balance is performed so that energy is balanced from one pipe to the next.

Does AFT Fathom Perform an Energy Balance Across Each Junction in the Model?

Yes, every junction except for heat exchangers and pumps are assumed to be adiabatic.

Does AFT Fathom Model Transient System Behavior?

Yes, in the optional [XTS Add-on Module](#).

Can I Model a Non-Newtonian Fluid in AFT Fathom?

Yes, AFT Fathom offers several [non-Newtonian fluid models](#) including [Power Law](#) and [Bingham Plastic](#).

Can AFT Fathom Model Fire Sprinkler Systems?

Yes. The Spray Discharge junction is perfectly appropriate for fire sprinklers. The fire sprinkler K values can be entered in directly into the Spray Discharge. In addition, Spray Discharge junctions can be turned on or off with a single mouse click. Evaluating multiple fire location scenarios is easily done, and use of the Scenario Manager allows all cases to be kept in the same model file.

Can AFT Fathom Model Ducting System Like Those in HVAC, Ventilation, and Dust Collection Systems?

Yes, AFT Fathom can [model ducts](#). The rectangular geometry is modeled as a non-cylindrical pipe using a hydraulic diameter. Since ducting systems are usually incompressible, the solution methodology is appropriate.

Can I Keep All of My Design Cases Together in a Single File?

Yes, the [Scenario Manager](#) lets you create dependent design cases where changes are inherited by children.

What Is the Difference Between Using a Junction to Input Pressure Loss Data and the Fittings & Losses in Pipes?

A junction has several advantages. First, solutions are given at all junctions, so the user can check the results at the junction. In contrast, Fittings & Losses are lumped into the pipe and it is not possible to give

results at the loss. Second, many junctions (such as valves) have the ability to specify a Restricted Flow Area for cavitation calculations. No such ability exists for Fittings & Losses; thus, cavitation is always ignored for Fittings & Losses. Third, when using a junction the location in the pipe system of the pressure loss is specified. In other words, the upstream and downstream pipe lengths are specified. In the case of Fittings & Losses, it assumed to act like a friction factor and be evenly distributed along the pipe. The Fittings & Losses approach has the advantage of being able to specify multiple losses quickly and easily, and not cluttering up the model Workspace with numerous junctions.

What Is the Difference Between a Reservoir Junction and an Assigned Pressure Junction?

There is an overlap in capability between the Reservoir junction and an Assigned Pressure junction and frequently they are interchangeable. Here are the differences. The Reservoir junction input pressure and temperature always corresponds to stagnation properties. In the Assigned Pressure junction, they can correspond to either static or stagnation properties. If static, only one pipe can be connected. If the stagnation option is used in the Assigned Pressure junction, it will behave identically to a Reservoir junction. Finally, the Reservoir junction allows pipes to connect at different elevations (i.e., depths), and also allows pipes to discharge above the liquid surface level.

How Does AFT Fathom Account for Pressure Losses at Tees and Wyes?

AFT Fathom uses sophisticated models to calculate losses at tees and wyes. The methods, which come from Idelchik, take into account losses that depend on the flow split, area change and angle of the branch pipe.

How Do I Size a Pump When There Are Not Any Flow Control Valves?

See the [Pump Sizing](#) topic.

How Do I Size a Pump When There Are Flow Control Valves?

See the [Pump Sizing](#) topic.

How Do I Model a Variable Speed Pump?

Enter the pump data as a polynomial in the Pump Properties Window and then set the pump speed directly. No entry is assumed to be 100% speed. If you enter something other than 100%, AFT Fathom uses the [Affinity Laws](#) to adjust the curve. If a control setpoint is entered, AFT Fathom will determine the speed necessary to meet that setpoint.

How Do I Find the Pump Speed to Obtain a Desired Flow or Pressure?

Enter the pump curve data in the Pump Properties Window and set the desired flow or pressure on the "Variable Speed" tab using the Controlled Pump option. AFT Fathom will calculate the pump speed required to produce the flow or pressure conditions specified and display the speed in the Pump

Summary report.

How Do I Close a Valve, Pump, or Pipe?

Select the pipe or junction you want to close and choose Special Condition from the Edit menu. By default, AFT Fathom will display a red "X" next to the pipe or junction label on the Workspace. It will also redraw your Workspace and show the closed sections of the model with dashed lines for the pipes and dashed outlines for each junction. Some special condition settings do not close the junction but have different purposes. For example, the normal condition for a relief valve is to be closed, so its special condition causes it to be open.

How Do I Model the Pressure Drop Across a Component When There Is Only One Data Point?

Use the [Resistance Curve Loss Model](#).

How Do I Model a Relief Valve in a System?

You can model a relief valve using either a Relief Valve junction or a regular Valve junction. The Relief Valve junction is always closed when you run the model (unless you set its Special Condition), and AFT Fathom will run the Solver to determine if sufficient pressure exists to crack it. If so, it will run the Solver again with the relief valve open. If you know the condition you are modeling will crack the relief valve, it is more efficient to just use a regular valve junction. In this case, AFT Fathom assumes the valve is open from the start, and does not have to go through the extra step of solving the network to check for sufficient system pressure to crack it.

How Do I Show the Pipe or Junction Names on the Workspace?

Open the pipe or junction Properties Window, click the Optional tab, and check or clear the check boxes for showing the number or name. This will affect the current pipe or junction. You can set the default behavior in the Workspace Preferences window available in [User Options](#).

How Do I Change the Display of Pipe or Junction Names on the Workspace After the Model Has Been Built?

You can use the Global Edit windows to change the current model settings for all pipes or junctions or only selected ones. See the Global Pipe Edit or Global Junction Edit window for more information.

How Do I Set My Preferred Engineering Units as Defaults?

From the Tools menu, select User Options. Then select Preferred Units under [Unit System](#). Drop down the list of units for each Unit Family to select the Preferred Unit. At this point, the preferred unit applies only to the current model. To make this the default for all models, click the Save as New User Defaults button at the bottom. The preferred units for a model can also be set in the Startup window.

How Do I Change the Input Data for Multiple Pipes or Junctions All at One Time?

The Global Pipe Edit and Global Junction Edit windows offer tremendous power and flexibility in changing all or parts of your model input all at once. See [Global Pipe Edit](#) and [Global Junction Edit](#) topics for more information.

How Do I Create a Liquid Mixture?

You can create mixtures with the [NIST REFPROP](#) or Chempak libraries (Chempak is an additional add-on). The AFT Standard fluid library does not support mixing. To create a mixture, go to the Fluid panel in Analysis Setup and choose either the Chempak or NIST REFPROP library, and then click the “Create New Mixture and Add...” button. Here you can specify the mixture components and percentages.

How Do I Merge Two Models Together?

Use the [Merge feature](#) on the File menu to merge models together.

How Do I Automatically Run Multiple Scenarios or Models Right After Each Other?

Multiple scenarios or models can be run sequentially using the Batch Run feature.

How Do I Enable or Disable the Highlight Feature in the Pipe and Junction Properties Windows?

There are three ways to enable or disable the Highlight feature. The first is toggling the option on the Options menu. The second is pressing the F2 function key while in a Pipe and Junction Properties Window. The third is double-clicking in the blank space at the top of the Pipe and Junction Properties Window.

How Do I Automatically Save My Results After the Model Is Run?

Open the Output Control window from the Analysis menu, change to the Format & Action tab, and choose to select the Transfer Results to Initial When Done, Transfer Valve States When Done, and Save Model When Results are Transferred options.

How Do I Quickly Find a Particular Pipe or Junction on the Workspace?

Use the Find feature to quickly find a pipe or junction.

How Do I Change the Reference Positive Flow Direction for One or More Pipes?

Select the pipe or pipes and choose the Reverse Direction feature on the Arrange menu.

How Do I Access Pipe and Junction Properties Windows from the Model Data Window?

In the Model Data tabular displays, double click on a row to automatically launch that pipe/junction's Properties Window.

How Do I Display Only Selected Pipes or Junctions in the Output?

Open the Output Control window from the Tools menu, change to the Show Pipes/Jcts tab, and select the pipes and or junctions you want to display.

How Do I Quickly Change the Units for an Output Parameter?

Whereas the Output Control window allows you specify units for all parameters, when in the Output window viewing results you can quickly change the units for parameters in the tabular displays by double-clicking the column header.

How Do I Model Elevation Changes in My Pipe System?

Set the elevations in the elevation fields for each junction. Pipes are assumed to be straight between junctions. If you need to model a system other than a stationary earth-based system, the gravitational acceleration can be changed on the Environmental Properties panel of Analysis Setup.

How Do I Show Data for Only Selected Pipes or Junctions in Workspace Layers?

This is controlled via the Label Control group of the [Layer Settings window](#).

How Do I Make the Pipe Drawing Tool Stay Active so That I Can Continue to Draw Pipes Without Having to Click It Each Time?

If you hold down the CTRL key when completing the pipe drawing (just before releasing the mouse button), the Pipe Drawing tool remains active, and you can draw a series of pipes without returning to the Toolbox each time. If you double-click the Pipe Drawing tool it remains active until you click it again a single time. This allows you to draw a series of pipes without returning to the Toolbox each time.

How Do I Add Graphical Segments to a Pipe so It Is Not Constrained to a Single Straight Line?

Use the Add/Remove Segments tool found on the Arrange Menu.

How Do I Set Up Custom Libraries on My Local PC?

See the [Libraries](#) topic.

How Do I Set Up Custom Libraries on Our Local or Wide Area Network?

See the [Network Libraries Overview](#) topic.

How Do I Change the Icon for a Junction in AFT Fathom?

Open the Junction Properties Window, click the Optional tab, and then click the Change Icon button.

Can I Customize the Junction Icons in AFT Fathom?

No, AFT Fathom icons are in a resource file that cannot be edited by the user.

What Is the Limit to the Size of Model I Can Create with AFT Fathom?

There are no theoretical limits to model size, but there are a few practical limits. First, AFT Fathom accepts pipe and junction ID numbers up to 30,000. This limits the model size to 30,000 pipes and 30,000 junctions. Before you reach that limit, however, you will likely encounter a limitation of your available RAM to hold all of the solver parameters. To determine how much RAM you need, add up the number of branches and tees in the model. Take the square of this number. Then multiply it by 32 to get the amount of RAM that must be available. For example, with 1,000 branches and tees, the square is 1 million, and after multiplying by 32 you need 32 million bytes of RAM (i.e., 32 MB).

Can AFT Fathom Model Reacting Flows?

No, AFT Fathom only models non-reacting flows.

Can I Install AFT Fathom Onto a Local or Wide Area Network?

Yes, AFT Fathom can run off the network or local PC. When installed on a network, the number of concurrent users is limited to the number of licenses purchased. See the [Information on Software Installation](#) topic.

Where Can I Find Reference Information About Pipe Materials?

See the [Pipe Material Library References](#) topic.

What Kind of Transients Can Fathom XTS Model?

A wide range, including liquid levels, valve open/close, pump speed changes, relief valve opening and more. Just as important, Fathom XTS can model time transients, event transients and inherent transients. Time transients are changes that are specified to occur based on the model time. Event transients, on the other hand, are triggered by an event that occurs in the system. Many kinds of events can be used, such as pressure, flow, velocity and liquid level. Inherent transients are changes that occur by the very nature of a system component. One example would be the liquid level change that occurs in a finite reservoir (or tank), while another would be a check valve opening and closing as a result of flow

reversals.

Related Topics

Overview of AFT Fathom.....	27
Engineering Assumptions.....	28
Fathom Learning Resources.....	63
Startup Window.....	64
New in Version 13.....	29
New in Version 12.....	40
New in Version 11.....	43
New in Version 10.....	48
New in Version 9.....	54

Fathom Learning Resources

Online Resources

Additional resources can be found in the [Learning Center](#) on our website. Resources found there include:

- **Training Seminar information**
- **Free Webinars**
- **Flow Expert Package information**
- **Case Studies**
- **Video Tutorials**
- **Tips & Tricks Blogs**
- **Quick Start Guides**

Online Quick Start Video Tutorial

The Quick Start video tutorial series provides a short introduction to the interface of AFT Fathom using a simple example model.

- [Online Quick Start Video Tutorial Series](#)

Beginner: Three Reservoir Problem

This is a simple example model which is recommended as a starting point for brand new AFT product users.

- [Beginner: Three Reservoir Problem](#)

Walk Through Examples

This links to a variety of example problems for beginning to advanced users which showcase different features and applications of AFT Fathom.

- [Walk Through Examples](#)

Module Walk Through Examples

A list of examples using the add-on modules of AFT Fathom to solve a variety of problems.

- [Module Walk Through Examples](#)

Related Topics

Overview of AFT Fathom.....	27
FAQ - Frequently Asked Questions.....	56
Engineering Assumptions.....	28
Startup Window.....	64
New in Version 13.....	29
New in Version 12.....	40
New in Version 11.....	43
New in Version 10.....	48
New in Version 9.....	54

Startup Window

The Startup Window is displayed when AFT Fathom is opened. This window allows the user to quickly access files and resources, as well as define a basic starting point for a new model.

- **Open a Model**
 - **My Files** - AFT Fathom keeps a list of recently opened model files. The most recent files can be opened with a single click here.
 - **Other Files** - Browse to any other model file, or open the built-in [Examples](#) folder.
- **Start a New Model**

- **Activate Modules** - Any licensed [Module](#) can be activated immediately. Checking these boxes will search for an available license.
- **Select Working Fluid**
 - **I Will Select Fluid Later** - No fluid will be defined, one will need to be defined in [Analysis Setup](#) on the [Fluid panel](#) before running the model.
 - **Use ASME Water** - Selects ASME Water to be used as the working fluid of the model which requires the definition of Pressure and Temperature, and optionally can have Heat Transfer modeling enabled.
 - **Select A Recently Used Fluid** - Allows the selection of a recently used fluid from the dropdown menu and also requires Temperature.
- **Modeling Preferences** - Displays various settings for the application, such as specifying the units to use, which [grid](#) to use, or whether to enable [NFPA](#). These options can be modified later. Click the **Modify** button to reveal the following settings, as shown in Figure 2. Click the **Remember My Preferences and Hide** button to update the modeling preferences for the model and to update the default settings for the application.
 - **Units** - US, Metric, or both sets of units with one as default can be displayed. Additionally, the displayed units can further be reduced to only show common units and those used by a certain industries.
 - **Language** - Select a language for the display of the Menus, [Workspace Layers](#), [Output](#), [Graph Results](#), and the Startup Window.
 - **2D or Isometric Workspace** - Set up the [grid](#).
 - **Default Pipe Material** - Set the [pipe material](#) that is initially selected for new pipes.
 - **NFPA Settings** - Enable [NFPA](#).
- **Resources**
 - **Current Version** - Open the AFT website to view the latest AFT Fathom release, or sign up for notifications of new releases via email.
 - **Learning Center** - Provides links to various [Learning Resources](#) that are designed to help get a new user up and running with AFT Fathom as quickly as possible. The resources include the Quick Start Guide PDF, [Examples](#), and video tutorials.

Overview of AFT Fathom

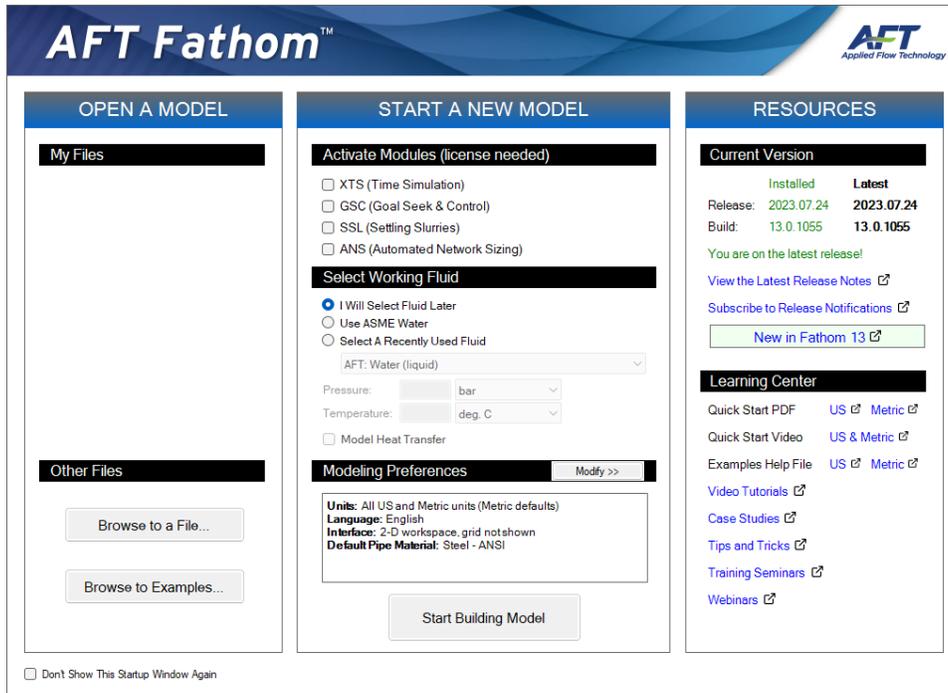


Figure 1: Startup Window in AFT Fathom

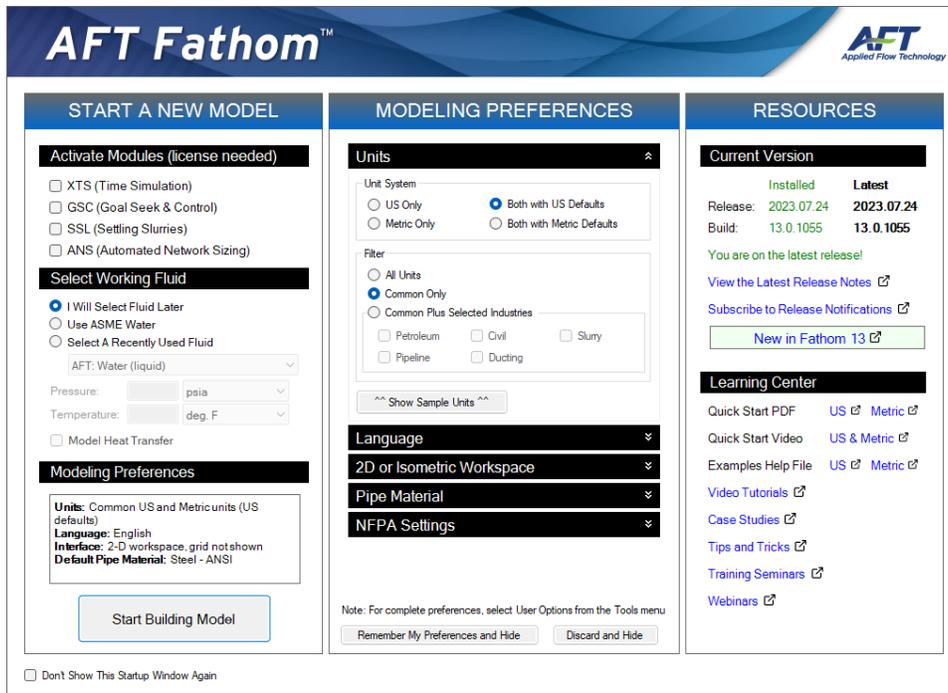


Figure 2: Startup Window showing Modeling Preferences

Related Topics

Overview of AFT Fathom.....	27
FAQ - Frequently Asked Questions.....	56
Engineering Assumptions.....	28
Fathom Learning Resources.....	63
New in Version 13.....	29
New in Version 12.....	40
New in Version 11.....	43
New in Version 10.....	48
New in Version 9.....	54

Fundamental Modeling Concepts

This section focuses on the critical concepts and theories that comprise the model building process in AFT Fathom. Topics include discussions on the role of pipe and junction objects, descriptions of how boundary conditions and flow direction are specified, and instructions on how to model closed systems.

Contents

Pipes and Junctions	68
Reference Positive Flow Direction	69
Features for Modeling Irrecoverable Losses	70
Philosophy of Computer Modeling	70
Role of Pressure Junctions and How They Work	72
Open vs. Closed Systems	75
Curve Fitting	79
Stagnation vs. Static Pressure Boundaries	80

Pipes and Junctions

AFT Fathom employs two fundamental pipe system constructs: pipes and junctions.

[Pipes](#) are conduits for incompressible, one-dimensional fluid flow. The flow rate through the entire length of the pipe is always constant in steady-state simulations, but can vary if the [XTS module](#) is enabled. Because pipes have constant diameters, the fluid velocity is also constant. [Pressure drop due to friction](#) occurs in pipes. [Heat transfer](#) may or may not occur depending on the type of model you select.

[Junctions](#) are connector points for pipes and are elements at which flow balances are made. Some junction types can only connect to one pipe; others can connect with up to twenty-five.

In addition to balancing flow, junctions also influence the flow or pressure behavior of the system. For example, a [reservoir junction](#) applies a constant pressure at a location, and the flow there is free to adjust in whatever manner is consistent with the governing equations. In contrast, an [assigned flow junction](#) applies a known flow rate at its location, allowing the pressure to adjust to that level dictated by the governing equations. The various junction types allow you to specify special kinds of [irrecoverable pressure losses](#) for fluid behavior.

Junctions communicate with each other through the pipes connecting them. Each pipe must be connected to two junctions. There are no exceptions to this rule.

A pipe differs from a junction in that it has a [reference positive flow direction](#). To say a pipe has a flow rate of 1 lbm/sec is meaningless unless the flow direction is specified.

In general, you do not need to specify the actual flow direction in a pipe, because AFT Fathom sorts out the true physical flow directions of the system you define. For example, if you specify the reference positive flow direction from left to right, and the solution is in fact the opposite direction, then that information will be presented in the output as a negative flow rate. Inlet and outlet conditions correspond to the reference positive flow direction, not the actual flow direction.

However, it is important to specify the proper flow direction for pipes which connect to junctions such as pumps or valves. AFT Fathom uses the flow direction of the pipes to determine in which direction to increase or reduce pressure.

Related Topics

Fundamental Modeling Concepts.....	68
Reference Positive Flow Direction.....	69
Features for Modeling Irrecoverable Losses.....	70
Philosophy of Computer Modeling.....	70
Role of Pressure Junctions and How They Work.....	72
Open vs. Closed Systems.....	75
Curve Fitting.....	79
Stagnation vs. Static Pressure Boundaries.....	80

Reference Positive Flow Direction

Reference Positive Flow Direction

Next to each pipe's ID number is an arrow that indicates the *reference positive flow direction* for the pipe. AFT Fathom assigns a flow direction corresponding to the direction in which the pipe is drawn. You can reverse the reference positive flow direction by choosing Reverse Direction from the [Arrange menu](#) or selecting the reverse direction button on the [Toolbar](#). With exception to some junctions, the reference positive flow direction is used for reference purposes only and does not need to correspond with the real direction of flow calculated by the Solver (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system). If the flow direction obtained by the Solver is opposite of the reference positive flow direction, then the output will show the flow rate as a negative number.

Note: The reference positive flow direction is important for pumps because the pipe directions will define which side is suction vs discharge. Furthermore, reverse flow through components may result in a [Caution message](#) which may or may not be valid depending on the intended direction of flow.

Convention for Flow Entering and Exiting

Many of the junction types permit fluid flow to come into or pass out of the system in a prescribed manner. The most obvious of these is the [Reservoir junction](#). Reservoirs maintain a constant pressure at a node, and the flow adjusts according to the governing equations. While in principle the fluid is stored or

removed from the reservoir, AFT Fathom considers the fluid as coming into or going out of the system. The stored reservoir fluid is not part of the system.

The convention for defining inward or outward flow for the system or any AFT Fathom object is that flow in is positive (that is, the object/system gains the flow) and flow out is negative (the object/system loses the flow).

Related Topics

Fundamental Modeling Concepts.....	68
Pipes and Junctions.....	68
Features for Modeling Irrecoverable Losses.....	70
Philosophy of Computer Modeling.....	70
Role of Pressure Junctions and How They Work.....	72
Open vs. Closed Systems.....	75
Curve Fitting.....	79
Stagnation vs. Static Pressure Boundaries.....	80

Features for Modeling Irrecoverable Losses

AFT Fathom works almost exclusively with the concept of the loss factor, also known as the K-factor or resistance factor. In some Properties windows, other popular methods for specifying loss information are provided where they are consistent with the loss factor method. For example, you will find Cd (discharge coefficients) for orifices.

AFT Fathom has methods available for modeling irrecoverable losses with the [Equivalent Length method](#) with either user-defined equivalent lengths or NFPA 13 table data (which is the table on the Fittings & Losses tab on the Pipe Properties window). In order for the fittings and losses to be modeled as equivalent lengths, the toggle switch must be set to use Equivalent Lengths. Refer to the [NFPA panel](#) topic for more information on generating an NFPA Output Report.

Related Topics

Fundamental Modeling Concepts.....	68
Pipes and Junctions.....	68
Reference Positive Flow Direction.....	69
Philosophy of Computer Modeling.....	70
Role of Pressure Junctions and How They Work.....	72
Open vs. Closed Systems.....	75
Curve Fitting.....	79
Stagnation vs. Static Pressure Boundaries.....	80

Philosophy of Computer Modeling

Some might say that a "good" computer model is one that accurately predicts the response of a system, but this is only partly true. The practicing engineer generally must weigh the accuracy of the model

against the risk, schedule, and/or budget involved. Through experience, an engineer learns where to focus resources and when an answer is good enough to base important decisions on.

Although AFT Fathom is designed to speed up the modeling process, it should never be used as a "black box." Your model input and output should always be carefully reviewed.

To make the most effective use of AFT Fathom, you should follow these guidelines:

- Be aware of what AFT Fathom can and cannot do and the [assumptions](#) it makes.
- Learn and use the tools in AFT Fathom that are appropriate for the specific analysis you want to perform. Detailed and accurate answers demand detailed and accurate input. Quick and dirty answers require less detailed input.
- Learn and use the tools in AFT Fathom that allow you to process information in the formats that mean the most to you. Because pipe flow applications vary so much within different industries, AFT Fathom is customizable for the way you want to do your pipe flow work. Understanding the tools will help you gain greater conceptual control over your pipe flow analyses and reduce modeling errors.
- Check your input data. Use the features available in AFT Fathom to review your results for input errors. Pay attention to the information in the output. Don't neglect warnings in the output. Perform hand analysis checks of results.
- Bound your problem. Instead of trying to calculate the exact resistance in a network, or the right loss factor for an orifice, use your judgment, guided by data, on how much variation might occur. By running your model at the upper and lower limits, you can obtain answers, for example, on the maximum and minimum possible pressure drops in the system. If the results at the maximum and minimum are acceptable, your analysis is probably finished. If they are not acceptable, you can further refine the input model.
- Recognize whether your problem is one of maximization or minimization. That is, recognize whether the design goal is to minimize the flow rate, for example, or maximize the flow rate. Then make your assumptions conservative enough to provide high confidence that your design requirements are being satisfied.

Related Topics

Fundamental Modeling Concepts.....	68
Pipes and Junctions.....	68
Reference Positive Flow Direction.....	69
Features for Modeling Irrecoverable Losses.....	70
Role of Pressure Junctions and How They Work.....	72
Open vs. Closed Systems.....	75
Curve Fitting.....	79
Stagnation vs. Static Pressure Boundaries.....	80

Role of Pressure Junctions and How They Work

Role of Pressure Junctions

In modeling a generalized pipe network, it is possible to construct models that do not have a unique solution. A common occurrence of this is when a model contains one or more sections which are completely bounded by known flow rates.

The simplest example is the system shown in Figure 1. In AFT Fathom terms, the system has two [assigned flow junctions](#).



Figure 1: Model with two assigned flows. This model does not have a unique solution.

Obviously, the flow in the pipe is known. But what is the pressure at the inlet? At the outlet? It cannot be determined because there is no *reference pressure*. The reference pressure is that pressure from which other pressures in the system are derived. There can be more than one reference pressure, but there always has to be at least one.

The model in Figure 1 can be built with AFT Fathom and if you try and run this model, AFT Fathom will inform you that it cannot run it because of the lack of a reference pressure (Figure 2). In AFT Fathom there are several junctions that can act as a reference pressure: the reservoir, assigned pressure, valve (with the exit valve option), orifice (with the exit orifice option), spray discharge, relief valve junctions which are open to an external pressure, and pressure control valves.

When you try and run a model that has one or more sections completely bounded by known flows (or Flow Control Valves or Pumps operating in fixed flow mode), the Need Reference Pressure window (shown below) is displayed. This window informs you which junctions bound the sections where a reference pressure is needed to obtain a unique solution.

In multi-pipe systems there are a host of other possibilities that present themselves. **All other configuration possibilities which lack a reference pressure ultimately boil down to the same problem that exists in Figure 1.**

See [Role of Pressure Junctions - Detailed Discussion](#) for more information.

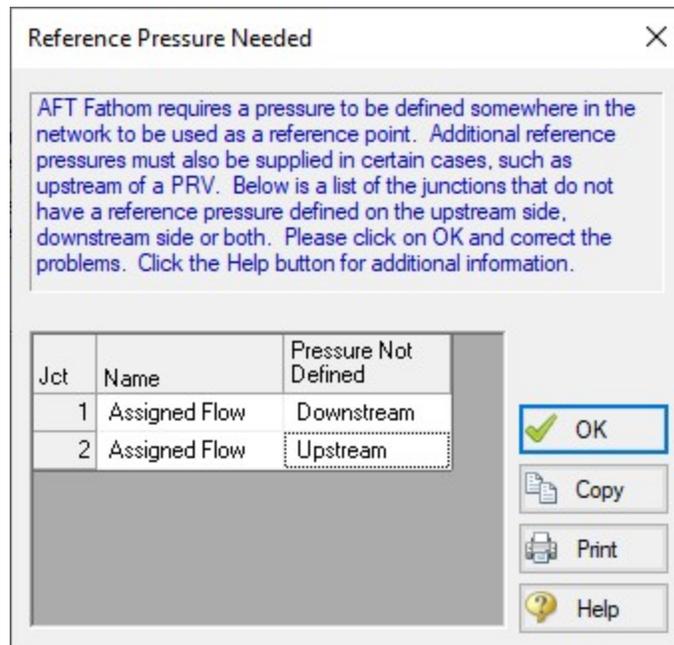


Figure 2: AFT Fathom message when you try to run a model without sufficient reference pressures

How Pressure Junctions Work

Pressure junctions (e.g., [reservoirs](#) or [assigned pressures](#)) in AFT Fathom are an infinite source or sink of fluid. This means that they can draw or discharge as much fluid as is necessary to maintain the specified pressure. This section offers a physical example of how pressure junctions work.

Consider the physical pumping system shown in Figure 3. This system pumps water from Lake Ontario to Lake Huron.

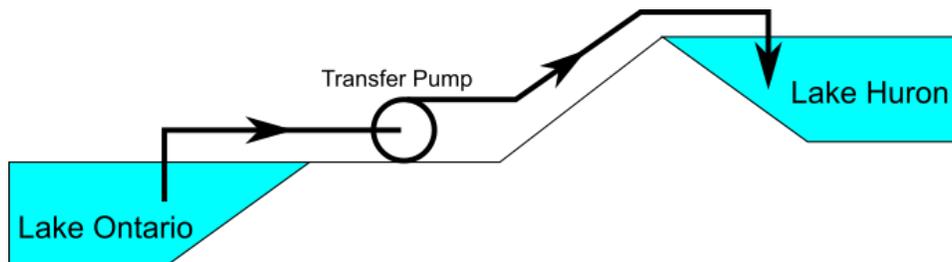


Figure 3: Physical pumping system to transfer water from shoreline supply tank (J1) to discharge tank (J2) at top of hill

An engineer is tasked with sizing the pump for this system. The engineer has an idea of the discharge head needed for the pump, and so builds an AFT Fathom model as shown in Figure 4. The Figure 4 model is attempting to represent the physical system in Figure 3, but it in fact represents the physical system in Figure 5.

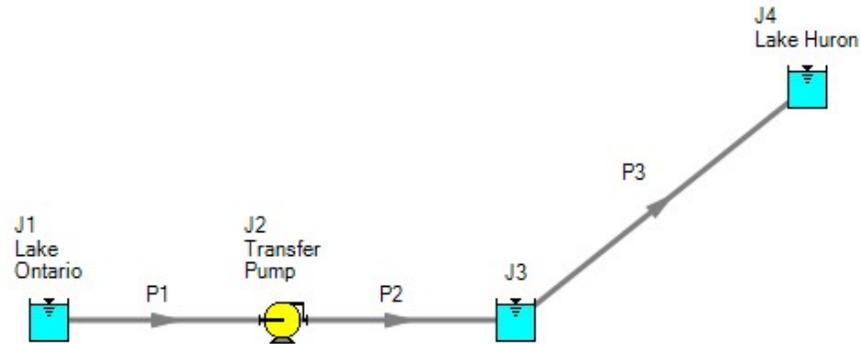


Figure 4: Model to size pump attempting to represent system in Figure 3. It does not accurately represent Figure 3

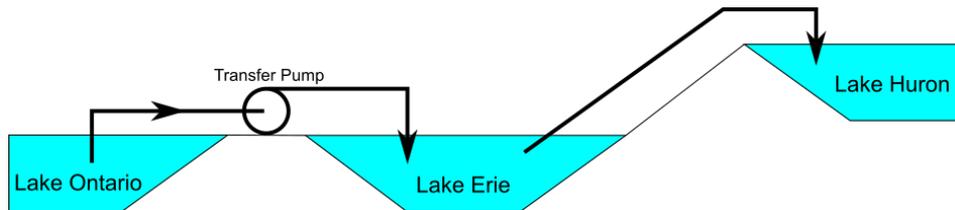


Figure 5: Physical system represented by model in Figure 4

Why? Because the reservoir junction (J3), which is an infinite source of fluid, has been located between the pump (J2) and Lake Huron (J4). The J3 reservoir isolates Lake Ontario and the pump (J1 and J2) from Lake Huron (J4). The J3 reservoir is very much like placing Lake Erie between the supply and discharge piping.

If the engineer changes the surface elevation in J3, it is like changing the surface elevation of Lake Erie. It will change the flow rate in the pipes, but no matter how high the water level of Lake Erie, it does not change the fact that the pumping system is isolated from Lake Huron. The flow rate in the piping from J4 to J3 is entirely dependent on the difference in surface elevations between J3 and J4, and is not influenced by the J2 pump in any way.

How then should the physical system in Figure 3 be modeled? It should be modeled as shown in Figure 6.

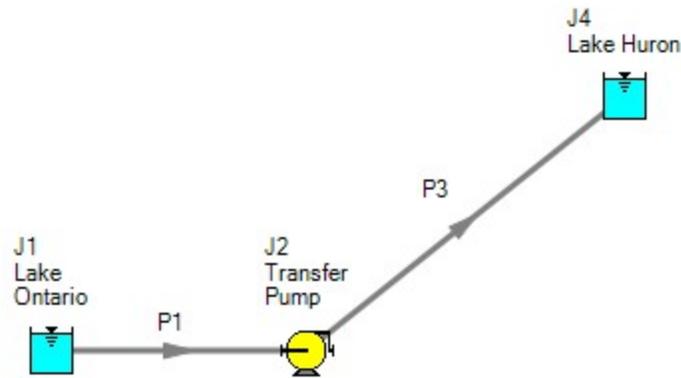


Figure 6: Proper model of physical system shown in Figure 3

If the goal is to size the pump at J2, the model in Figure 6 can be used with the pump modeled as a fixed flow pump.

Related Topics

Role of Pressure Junctions - Detailed Discussion	755
Fundamental Modeling Concepts	68
Pipes and Junctions	68
Reference Positive Flow Direction	69
Features for Modeling Irrecoverable Losses	70
Philosophy of Computer Modeling	70
Open vs. Closed Systems	75
Curve Fitting	79
Stagnation vs. Static Pressure Boundaries	80

Related Examples

Pump Sizing and Selection with Flow Control Valves	882
--	-----

Open vs. Closed Systems

Model a Closed Loop System

A closed circulating system can easily be modeled with AFT Fathom by locating a [Reservoir junction](#) anywhere in the system and setting the pressure to the known pressure at that point. Inflow and outflow pipes will then need to be connected. The Reservoir junction merely acts as a "reference pressure" against which all other pressures in the system are compared. Because there are no other inlets or

outlets to the system, then whatever is delivered by the Reservoir junction to the system will also be received from the other pipes, thus achieving a net zero outflow rate for the system.

When using a Reservoir junction as a reference pressure for a closed system, you should select the Balance Energy in Reservoir option, which will allow the reservoir temperature to vary and be calculated in a manner consistent with the overall energy balance.

Balancing Mass

In order to model a closed system, only one pressure junction is used in the model. Typically this would be either a Reservoir or Assigned Pressure. Pressure type junctions are an infinite source of fluid and do not balance flow. How then can one be used to balance flow in a closed system?

To answer this question, it is worth considering how AFT Fathom views a closed system model. AFT Fathom does not directly model closed systems, and in fact does not even realize a closed system is being modeled.

Consider the system shown in Figure 1. This is an open system. Fluid is taken from J1 and delivered to J4 and J5. Because AFT Fathom's solution engine solves for a mass balance in the system, all flow out of J1 must be delivered to J4 and J5. Because the flow is steady-state, no fluid can be stored in the system; what goes in must go out.

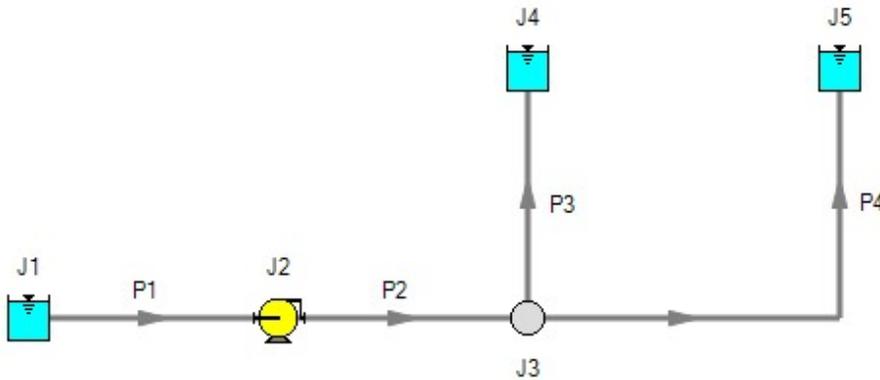


Figure 1: Open system - Flow out of J1 equals the sum of J4 and J5

Now consider the systems in Figure 2. The first system appears to be closed, while the second appears to be open. If the same boundary condition (i.e., surface elevation and pressure) is used for J1, J11 and J12 in the second system, it will appear to AFT Fathom as an identical system to the first system. The reason is that AFT Fathom takes the first system and applies the J1 reservoir pressure as a boundary condition to pipes P4, P9 and P10. The second system uses three reservoirs to apply boundary conditions to P4, P9 and P10. But if the reservoirs all have the same elevation and pressure, the boundary conditions are the same as J1 in the first system. Thus the same boundary condition is used for P4, P9 and P10 in both models, and they appear identical to AFT Fathom.

But how is the flow balanced at J1 in the first system? Looking at the first system, one sees that to obtain a system balance, whatever flows into P10 must come back through P4 and P9. Because there is overall system balance by the solver, it will give the appearance of a balanced flow at the pressure junction J1. If there is only one boundary (i.e., junction) where flow can enter or leave the pipe system, then no flow will

enter or leave because there isn't anywhere for it to go. Thus the net flow rate will be zero at J1 (i.e., it will be balanced). But recognize that AFT Fathom is not applying a mass balance to J1 directly. It is merely the result of an overall system balance.

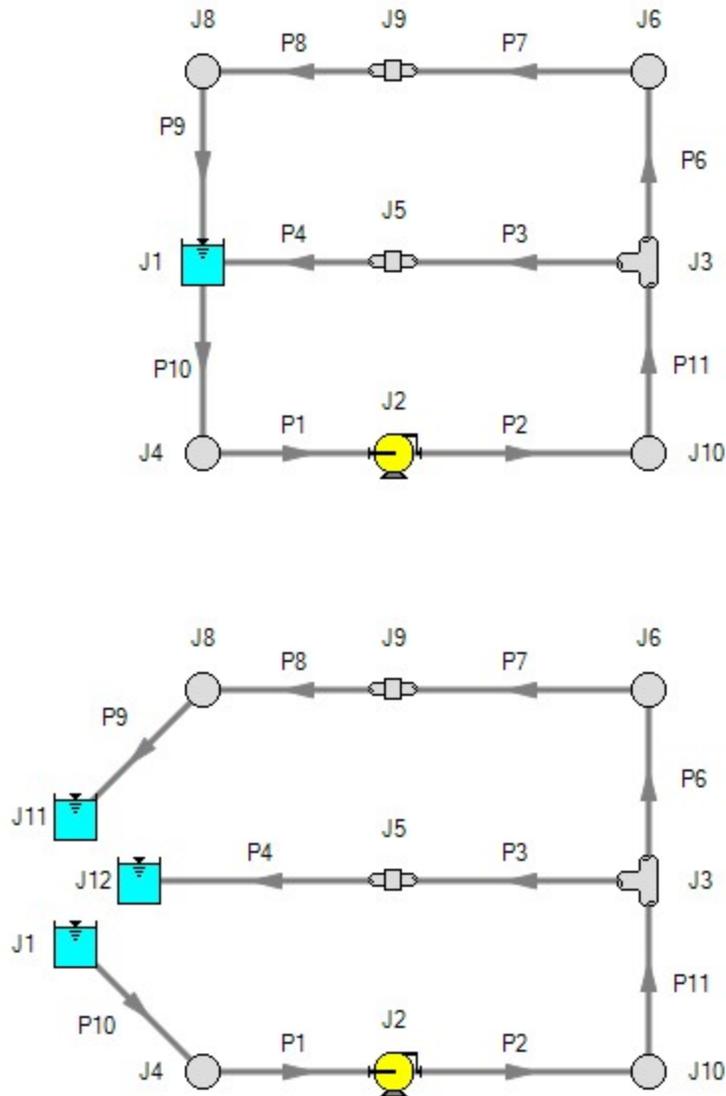


Figure 2: The first system is closed, and the second open. In both systems the flow into P10 is the same as sum of P4 and P9. If the second system has the same conditions at J1, J11 and J12, the two system will appear identical to AFT Fathom.

Balancing Energy

Here the thermal aspects of open and closed systems are discussed.

Curve Fitting

There are many places within the interface where the user may be asked to define how a property changes in relation to another. Some examples include:

- [Pump Curves](#)
- [Junction Resistance Curves](#)
- [Custom Fluids](#)
- [Pipe Physical Properties](#)

These properties are entered in a tabular format, relating a particular property to one or several others. For example, see the below definition of a custom fluid:

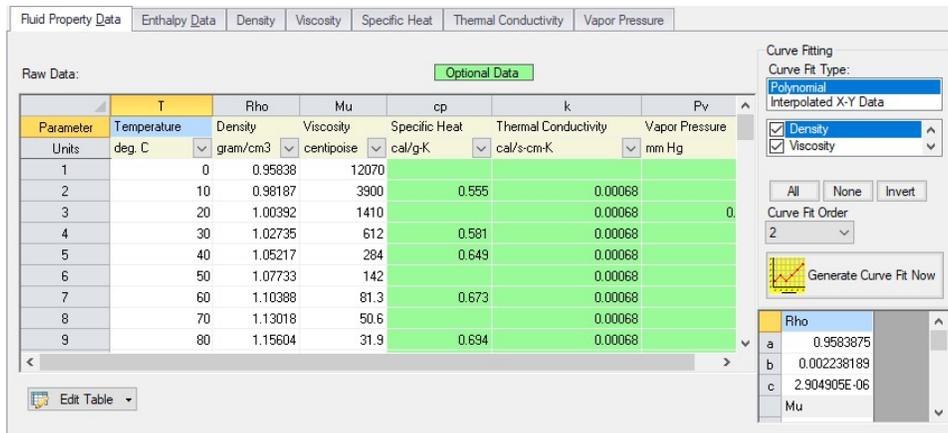


Figure 1: Custom Fluid Definition

There are several parameters that control the behavior of the fluid and they are entered as they relate to temperature.

However, it is important to recognize that these values in particular are not directly used in the solution. Instead, the data is used to create a best-fit curve. This curve is then used in the solution process. This is an important step to take because there must be a continuous relationship between the independent variable (e.g. temperature) and the dependent variable (e.g. density). Even if the final solution was close to one of the defined points, the Solver needs continuous data to proceed.

There are some options for how the data is curve fit. First, there are two Curve Fit Types:

- **Polynomial** - The data is best fit to a polynomial of order 0, 1, 2, 3, or 4. Note that a 0th order polynomial is a constant value, and a 1st order is a linear fit.
- **Interpolated X-Y Data** - The data has a linear interpolation drawn between every data point.

One of the critical items to note with any curve fit is that you may not be operating close to one of your defined points if the data does not conform well to a polynomial relationship. For example, the above fluid has a rapidly decaying viscosity that does not follow a polynomial relationship. Fitting the data to the standard 2nd order polynomial results in the following:

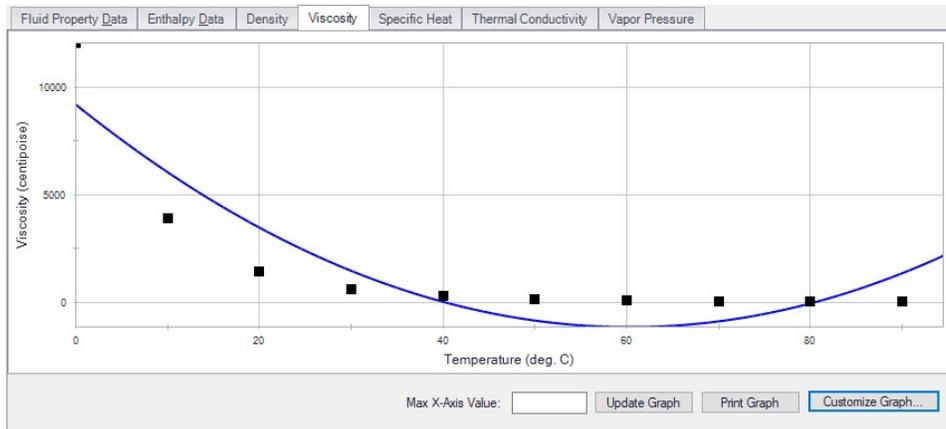


Figure 2: Using a Polynomial Curve Fit on Non-Polynomial Data

The viscosity relationship in the model will use the blue curve, and not the user-defined black points. Depending on the operating range of the system, this may significantly impact the results! For this data in particular, using the Interpolated X-Y Data option may be a better choice.

After data has been curve fit, the constants for the resulting curve are displayed in the lower-right of the associated window.

Related Topics

Fundamental Modeling Concepts.....	68
Pipes and Junctions.....	68
Reference Positive Flow Direction.....	69
Features for Modeling Irrecoverable Losses.....	70
Philosophy of Computer Modeling.....	70
Role of Pressure Junctions and How They Work.....	72
Open vs. Closed Systems.....	75
Stagnation vs. Static Pressure Boundaries.....	80
Flow solution for Junction X is above/below the maximum/minimum value specified for the resistance curve. Flow rate: Y.....	785

Stagnation vs. Static Pressure Boundaries

With two exceptions (to be discussed), all pressure-type boundary conditions in AFT Fathom are stagnation. This works very well for things such as storage tanks, cooling ponds, lakes, etc., where the volume associated with a pressure is large and will not change (significantly) with time. These boundaries have no velocity associated with them, and using stagnation pressure is thus appropriate. These boundary conditions are most clearly rendered in AFT Fathom by use of a [Reservoir junction](#).

Consider the reservoir and connected pipe in Figure 1. When the liquid in a reservoir flows into connecting pipes, the static pressure immediately drops due to the increase in velocity.

It is tempting to say that because there is no velocity in the reservoir, then it does not matter whether the reservoir boundary pressure is considered as a static or stagnation pressure because they are equal. This is a misconception. The pressure boundary condition in an AFT Fathom model is applied at the exit of the reservoir, not within the reservoir itself. Inside the reservoir, the static and stagnation pressures are the same, but they are not the same at the exit - they differ by the amount of the dynamic pressure.

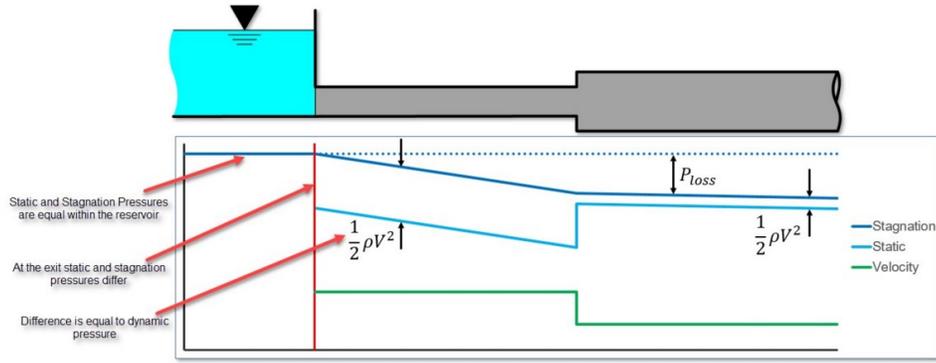


Figure 1: Pressure profile along horizontal pipe shown

The choice of either stagnation or static pressure as a boundary condition depends on whether there is a velocity associated with the pressure that is specified. If the pressure is in a vessel or large (lower velocity) header, then a stagnation pressure is appropriate. If the pressure is in a pipe somewhere in the middle of a system, static pressure is more suited.

Pressure boundary conditions with no velocity are more common across all piping industries, and are usually appropriate. However, this is not always the case.

When to use Static Pressure

Now let's consider the application of a true static pressure boundary condition. A static boundary condition has a unique velocity associated with it. The Bernoulli equation, which in this integrated form applies only to incompressible flow, is given by:

$$P_1 + \frac{1}{2} \rho V_1^2 + \rho g z_1 = P_2 + \frac{1}{2} \rho V_2^2 + \rho g z_2 + \Delta P_f$$

From this equation it can be seen that application of the static pressure is not sufficient to specify location 1. The velocity and elevation is also required.

In AFT Fathom, elevation data is entered for junctions. The pipes adopt the elevation of the junction to which they are connected.

But where does the velocity information for the Bernoulli equation come from? One might respond that it can be obtained from the flow rate. But this raises another question: what if the flow rate isn't known? Put another way, what if the flow rate is what we are solving for?

When the user supplies a stagnation pressure boundary condition, AFT Fathom can use Equation 6 in the [Network Solution Methodology](#) topic. Here, no velocity information is required. A stagnation pressure

boundary can thus be supplied to multiple pipes that might have different velocities. That is why the same reservoir junction can connect with multiple pipes.

However, if the user specifies a static pressure as the boundary condition, to apply Equation 4 in the [Network Solution Methodology](#) topic, a unique velocity must be supplied. Thus, it is only possible to connect a static pressure to a single pipe with a unique velocity.

Important: A static pressure boundary has a unique velocity associated with it, and can thus connect to only one pipe.

Where would one find a static pressure boundary? The best example is when the boundary condition is inside a pipe. A pipe system model can start and end anywhere it is convenient for the user. It may be convenient to not start the model at the physical boundary (such as a tank) but at a particular location in the pipe system. This could be, for example, at the location of a pressure measurement. Or it could be at the boundary of the pipe system for which your company is responsible, with another company responsible for what is on the other side of that boundary.

If one needs to model such a situation, the [Assigned Pressure junction](#) allows one to model either a static or stagnation pressure. The default stagnation pressure allows [connection of up to 25 pipes](#). If static pressure is chosen, only one connecting pipe is allowed.

Static Pressure at Pressure Control Valves

Another example of static pressure is at pressure control valves. For [pressure control valves](#) (i.e., PRV's and PSV's) the default control pressure is static pressure. The reason is that the measured pressure that provides feedback to the controller will typically be a static pressure measurement. You have the option of modeling pressure control valves as either static or stagnation pressure.

Related Topics

Fundamental Modeling Concepts.....	68
Pipes and Junctions.....	68
Reference Positive Flow Direction.....	69
Features for Modeling Irrecoverable Losses.....	70
Philosophy of Computer Modeling.....	70
Role of Pressure Junctions and How They Work.....	72
Open vs. Closed Systems.....	75
Curve Fitting.....	79

Analysis Setup Overview

Analysis Setup contains all of the settings which the user must define to run the model, including fluid properties, pipe and junction status, environmental properties, and other related options. Open Analysis Setup from either the Common Toolbar or the [Analysis menu](#).

The controls in Analysis Setup are divided into different *groups* in the navigation pane on the left side of the window (Figure 1). Each group has one or multiple *items*. Clicking on an item opens the *panel* for that item.

A group is not fully defined until each item in that group is defined. Each group will have either a green check mark, indicating that all required items have been defined, or a red exclamation point, indicating that some or all of the required inputs are missing. Note that some groups will have a green checkmark by default, indicating that the required inputs have default settings already applied, or all inputs for that group are optional. Groups that are undefined will change to a green check mark when the required inputs for that group are entered, and the user navigates to a different panel. Note that none of the changes will be applied to the model until OK is selected. Clicking Cancel will discard all changes made in the window.

The groups in Analysis Setup are listed below:

- [Modules Group](#)
- [Fluid Properties Group](#)
- [Pipes and Junctions Group](#)
- [Transient Control Group](#) - ([XTS Module](#) only)
- [Goal Seek and Control Group](#) - ([GSC Module](#) only)
- [Steady Solution Control Group](#)
- [Environmental Properties Group](#)
- [Cost Settings Group](#)
- [Automatic Sizing Group](#) - ([ANS Module](#) only)
- [Miscellaneous Group](#)

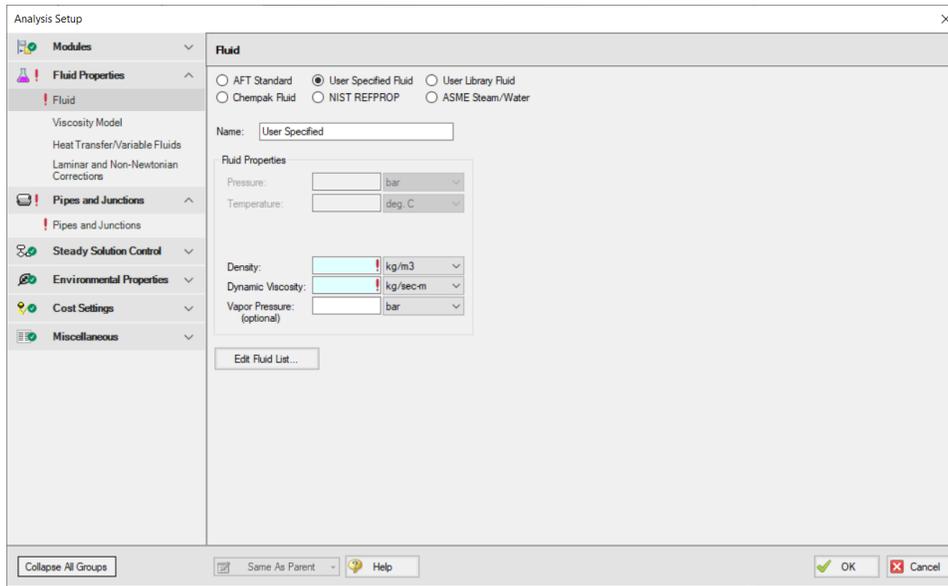


Figure 1: Default state of Analysis Setup in a new blank model

Related Examples

Beginner - Three Reservoir823

Modules Group

AFT Fathom has four optional add-on modules which extend its modeling capabilities into other areas. The modules allow multivariable goal seeking and control simulation, settling slurry modeling, time varying simulation, and full system sizing. The [Modules panel](#) within the Modules group in [Analysis Setup](#) allows you to activate and enable any of the four AFT Fathom add-on modules. An additional license to the base AFT Fathom license is required to use a module. These modules include:

- [GSC \(Goal Seek & Control\)](#) - Identifies input parameters that yield desired output values and simulates control functions.
- [SSL \(Settling Slurry\)](#) - Models the effects of pumping fluids containing settling solids using the Wilson/GIW method.
- [XTS \(Extended Time Simulation\)](#) - Models dynamic system behavior and how critical system parameters vary over time.
- [ANS \(Automated Network Sizing\)](#) - Evaluates the complex interaction of system variables to reveal combinations of cost savings.

Related Topics

Modules Panel..... 85

Add-on Modules.....	538
Analysis Setup Overview.....	83
Fluid Properties Group.....	90
Pipes and Junctions Group.....	141
Transient Control Group.....	143
Goal Seek and Control Group.....	148
Steady Solution Control Group.....	162
Environmental Properties Group.....	168
Cost Settings Group.....	170
Automatic Sizing Group.....	173
Miscellaneous Group.....	175

Modules Panel

The Modules panel allows users to activate and enable the optional add-on modules.

Activating a module checks out a license seat for the module.

Enabling a module turns on all functionality for the module. By default activating a module will also automatically enable the module for use. Disabling a module keeps the license seat checked out but the module calculations are not applied to the model. Disabling a module may be desired for various reasons, as is discussed further for each module below.

Note: A module specific license is required to activate the GSC, SSL, XTS, or ANS Add-on Modules.

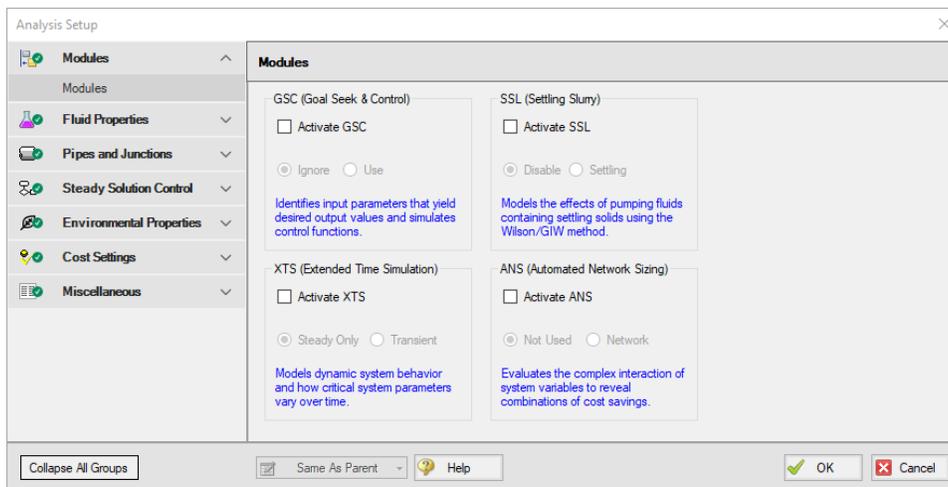


Figure 1: The default state of the Modules panel

GSC (Goal Seek & Control)

Activate the Goal Seek & Control module by checking the box next to Activate GSC. Once activated, enable it by selecting Use or disable it by selecting Ignore. When the GSC module is activated, the [Goal Seek and Control Group](#) appears in the Analysis Setup window. The Goal Seek and Control group has three items: [Variables](#), [Goals](#), and [Numerical Controls](#).

The GSC module can be enabled before or after a model is built. Pre-existing models built with the base AFT Fathom application can have the GSC module activated.

Setting the GSC module to Ignore causes the GSC module to function like the base AFT Fathom application. One difference is that users are still able to enter goal seeking data with this data retained in the model. If the model is opened in the base AFT Fathom application, this data will be lost. Table 1 relates the differences between using AFT Fathom without the GSC module activated, using it with the GSC module activated but disabled (Ignore mode), and using it with the GSC module activated and enabled.

The GSC module can also be toggled between Use and Ignore from the Analysis menu in the Goal Seek & Control option as shown in Figure 2. Use this feature to quickly toggle the GSC module between Use and Ignore without opening the Modules panel in Analysis Setup.

Table 1: AFT Fathom feature accessibility based on the GSC module activation and enablement

Features	GSC Not Activated	GSC Activated	
		GSC Disabled	GSC Enabled
Goal Seek and Control Group in Analysis Setup	Not Visible	Visible	Visible
Goal Seek and Control in Analysis menu	Not Visible	Visible	Visible
Goal Seek and Control tab in General Section of Model Data window	Not Visible	Visible	Visible
GSC Variables and GSC Goals tabs in General Section of Output window	Not Visible	Not Visible	Visible

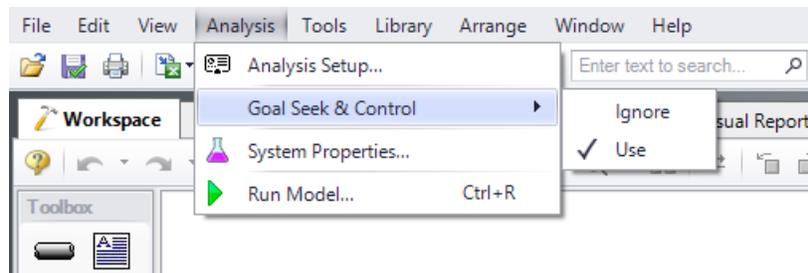
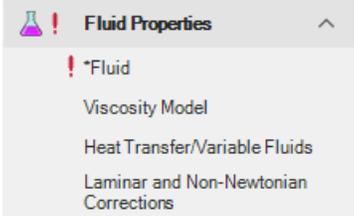


Figure 2: Select Use from the Goal Seek & Control option in the Analysis menu to perform goal seeking when running the model

SSL (Settling Slurry)

Activate the Settling Slurry module by checking the box next to Activate SSL. Once activated, enable it by selecting Settling or disable it by selecting Disable. When the SSL module is activated, the four standard items in the Fluid Properties group are replaced with the three SSL module fluid properties items as shown in Table 2 below. These items are [Carrier Fluid](#), [Solids Definition](#), and [Slurry Definition](#). Additionally, the Library menu will have a Solids Library option. These items are present while SSL is activated and enabled, but will be switched back if SSL is disabled.

Table 2: Analysis Setup items in Fluid Properties group with and without SSL active

Without SSL Active	With SSL Active
 <p>The screenshot shows the 'Fluid Properties' menu with the following items: *Fluid, Viscosity Model, Heat Transfer/Variable Fluids, and Laminar and Non-Newtonian Corrections.</p>	 <p>The screenshot shows the 'Fluid Properties' menu with the following items: *Carrier Fluid, Solids Definition, and Slurry Definition.</p>

Activating the SSL module can be done before or after a model is built. If activated after, then additional input will be required for solids data. Pre-existing models built with standard AFT Fathom can be opened with SSL and settling slurry data added.

Settling slurry calculations can be turned off at any time by selecting Disable. The Disable mode causes the SSL module to function like standard AFT Fathom. If a model with SSL activated is opened in standard AFT Fathom without SSL activated, then the SSL data will be lost once the model is saved. Table 3 relates the differences between not using SSL and using it in Disable mode. The purpose of Disable mode is that it allows standard clear fluid calculations to be performed without deactivating SSL. Deactivating the SSL module will cause the loss of the slurry input data for the active scenario.

The SSL module can also be toggled between Settling and Disable from the Analysis menu in the Slurry option as shown in Figure 3. Use this feature to quickly toggle the SSL module between Settling and Disable without opening the Modules panel in Analysis Setup.

Table 3: AFT Fathom feature accessibility based on the SSL module activation and enablement

Features	SSL Not Activated	SSL Activated	
		SSL Disabled	SSL Enabled
SSL items in Fluid Properties group	Not Visible	Not Visible	Visible
Slurry in Analysis menu	Not Visible	Visible	Visible
Solids Library in Library menu	Not Visible	Visible	Visible
Flowrate specified in terms of slurry, solids or clear liquid in several junctions	Not Visible	Not Visible	Visible
Slurry data in Output	Not Visible	Not Visible	Visible
Initial Velocity Guess Ratio V/Vsm in Advanced Parameters panel	Not Visible	Not Visible	Visible

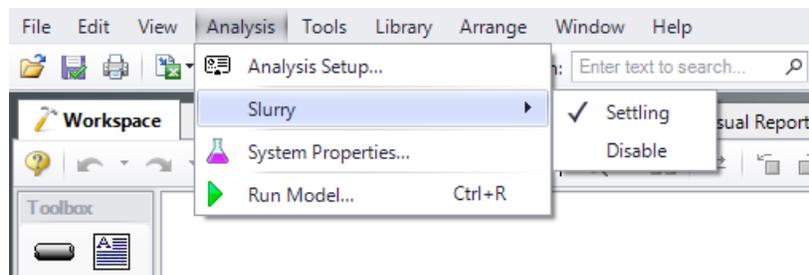


Figure 3: Select Settling from the Slurry option in the Analysis menu to perform settling slurry calculations when running the model

XTS (Extended Time Simulation)

Activate the Extended Time Simulation module by checking the box next to Activate XTS. Once activated, enable it by selecting Transient or disable it by selecting Steady Only. When the XTS module is activated, the [Transient Control group](#) appears in the Analysis Setup. The Transient Control group has one item: [Simulation Duration](#).

Activating the XTS module can be done before or after a model is built. If activated after, then additional input will be required for transient junctions. Pre-existing models built with the base AFT Fathom application can have the XTS module activated.

Transient modeling can be turned off at any time by selecting Steady Only. The Steady Only mode causes the XTS module to function like standard steady-state AFT Fathom. One difference is that users can still enter transient data and this data is retained in the model. If the model is opened in standard AFT Fathom, this data will be lost. Additionally, the Transient Control group and Simulation Duration panel are still accessible. Table 4 relates the differences between not using XTS and using it in Steady Only mode.

The XTS module can be toggled between Steady Only and Transient from the Analysis menu in the Time Simulation option as shown in Figure 4. Use this feature to quickly toggle the XTS module between Steady Only and Transient without opening the Modules panel in Analysis Setup.

Table 4: AFT Fathom feature accessibility based on the XTS module activation and enablement

Features	XTS Not Activated	XTS Activated	
		XTS Disabled	XTS Enabled
Transient Control Group in Analysis Setup	Not Visible	Visible	Visible
Time Simulation in Analysis Menu	Not Visible	Visible	Visible
Transient tab in junction properties windows	Not Visible	Visible	Visible
Transient Pipe and Jct tabs in Output	Not Visible	Not Visible	Visible
Transient tab in junction section of Model Data	Not Visible	Visible	Visible
Time control slider and buttons in Output	Not Visible	Not Visible	Visible
Time control slider and buttons in Workspace	Not Visible	Not Visible	Visible
Transient tab on Select Graph Data	Not Visible	Not Visible	Visible
Animation controls in Graph Results	Not Visible	Not Visible	Visible

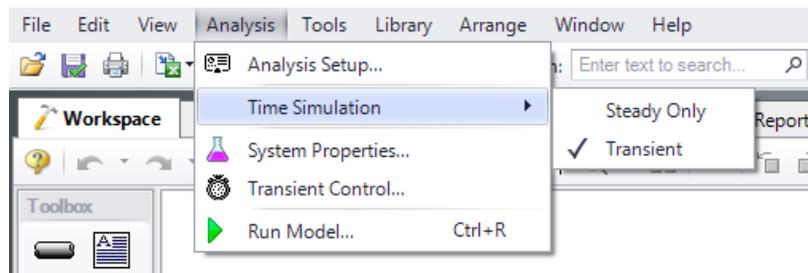


Figure 4: Transient mode is enabled from the Analysis menu

ANS (Automated Network Sizing)

Activate the Automated Network Sizing module by checking the box next to Activate ANS. Once activated, enable it by selecting Network or disable it by selecting Not Used. When the ANS module is activated, the [Automatic Sizing group](#) appears in the Analysis Setup. The Automatic Sizing group has one item: [Sizing Summary](#). Additionally, the [Sizing Window](#) becomes available when ANS is activated.

Activating the ANS module can be done before or after a model is built. If activated after, then additional sizing data will be required to run the model. Pre-existing models built with the base AFT Fathom application can have the ANS module activated.

ANS can be disabled at any time by selecting Not Used from the same menu. The Not Used mode causes the ANS module to function like the base AFT Fathom application. However, all ANS data and settings in the model will still be retained. If the model is opened in the base AFT Fathom application, this data will be lost. The purpose of Not Used mode is that it allows ANS to be quickly turned off and on without having to change data in the [Sizing Objective](#).

The ANS module can be toggled between Network and Not Used from the Analysis menu in the Automated Sizing option as shown in Figure 5. Use this feature to quickly toggle the ANS module between Network and Not Used without opening the Modules panel in Analysis Setup.

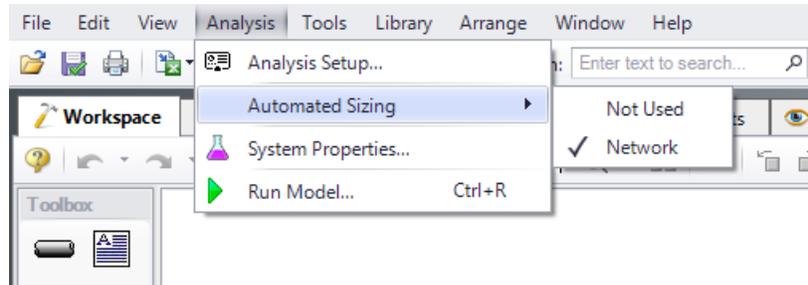


Figure 5: Enabling Automated Sizing from the Analysis menu

Related Topics

Modules Group.....	84
Add-on Modules.....	538

Fluid Properties Group

The Fluid Properties group in [Analysis Setup](#) gives you control over the fluid properties that influence your pipe system analysis.

While there are a four panels available to input the fluid properties, only one panel is required to run the model. The rest of the panels offer optional input.

Note: If the SSL module is activated the panels in the Fluid properties group will change to show the slurry panels. See [SSL Module Fluid Properties](#) for more information.

- [Fluid panel](#) - Required panel. Allows the user to define the fluid properties directly, or from one of the available fluid libraries.
- [Viscosity Model panel](#) - Optional panel. Allows user to define a non-Newtonian viscosity model.
- [Heat Transfer/Variable Fluids panel](#) - Optional panel. Allows user to enable one of the heat transfer property options.
- [Laminar and Non-Newtonian Corrections panel](#) - Optional panel. Allows the user to choose what correction factors to apply for laminar or non-Newtonian flow and to which objects the corrections should apply.

Related Topics

Fluid Panel.....	91
Viscosity Model Panel.....	97
Heat Transfer/Variable Fluids Panel.....	120
Laminar and Non-Newtonian Corrections Panel.....	123
SSL Module Fluid Properties Group.....	131
Analysis Setup Overview.....	83
Modules Group.....	84
Pipes and Junctions Group.....	141
Transient Control Group.....	143
Goal Seek and Control Group.....	148
Steady Solution Control Group.....	162
Environmental Properties Group.....	168
Cost Settings Group.....	170
Automatic Sizing Group.....	173
Miscellaneous Group.....	175

Fluid Panel

The Fluid panel controls the properties of the fluid in the model. A default fluid is not selected - one must be defined to run any simulation unless one was previous defined on the [Startup Window](#).

The first selection that must be made is what Fluid Library to use. The default selection of User Specified requires the manual definition of Density, Dynamic Viscosity, and Vapor Pressure (optional if cavitation detection is not required). The available [Fluid Libraries](#) are:

- [AFT Standard](#) - Common fluids from various industry standard sources have been curve fit to optimize calculation speed. This fluid library offers the best balance between accuracy and computation time. Many of these fluids have thermal property information that allows for the calculation of [heat transfer](#).
- [ASME Steam/Water](#) - Properties are calculated via the Equations of State which were used to generate the classic steam/water tables in the IAPWS Industrial Formulation (1997).
- [NIST REFPROP](#) - The Reference Fluid Thermodynamic and Transport Properties (REFPROP) Database is licensed from the National Institute of Standards and Technology (NIST) where properties for approximately 150 fluids are calculated via Equations of State. REFPROP supports user-specified fluid mixtures. For more info see the following sources: [Program Summary](#), [REFPROP FAQ](#), [NIST Chemistry WebBook](#), [NIST Chemical Name Search](#).
- [Chempak Fluid](#) - Optional add-on for purchase which contains data for approximately 700 fluids and supports user-specified fluid mixtures. Chempak is licensed from Madison Technical Software.
- **User Library Fluid** - If the desired working fluid is not available via the other sources, custom fluid properties can be entered and saved to the User Library for repeated use in the future. Fluids can be created using the [Fluid Libraries](#) menu.

Select a library by clicking the radio button next to the name of the library. After selecting a library, the Fluids Available in Library list is populated. One of the fluids can be selected and used with the Add To Model button.

Note: All fluids in Fathom are treated as incompressible as stated in the [Engineering Assumptions](#). This includes fluids in the gas phase and fluids above the critical point.

AFT Standard Fluid Library

Common fluids from various industry standard sources have been curve fit to optimize calculation speed. This fluid library offers the best balance between accuracy and computation time. Many of these fluids have thermal property information that allows for the calculation of [heat transfer](#).

Generally, fluids in the AFT Standard library are assumed to have properties that only vary with temperature which follows the incompressible engineering assumption of Fathom.

To add a fluid into your model, select it from the Fluids Available in Library list and click the Add to Model button. The fluid will appear in the Fluids in Current Model area. Once the fluid is added, specify the temperature in the Fluid Properties area on the right to calculate values of density, dynamic viscosity, and vapor pressure, as shown in Figure 1.

Vapor pressure is listed as optional since it is only needed to check if cavitation exists in the pipes/junctions.

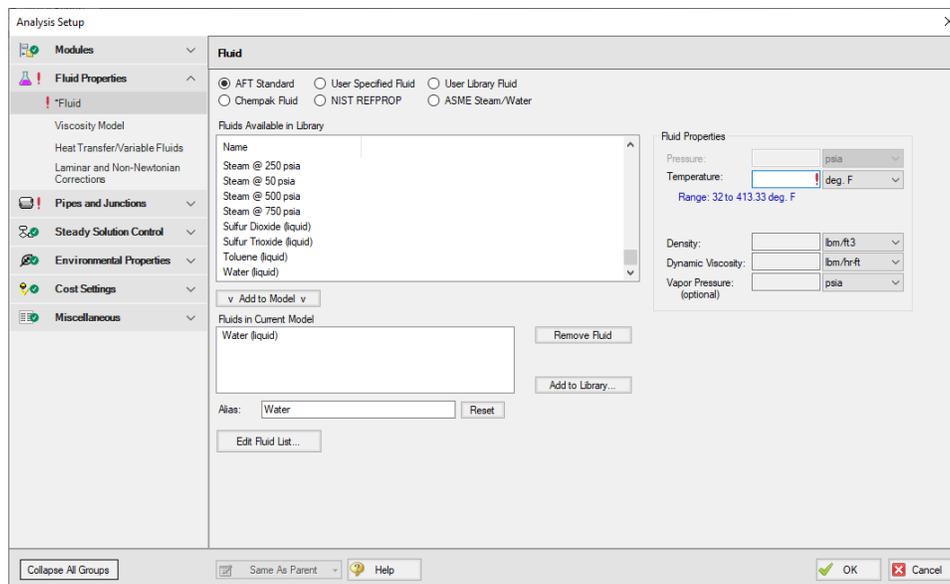


Figure 1: Fluid panel in Analysis Setup with the AFT Standard fluid library selected

ASME Steam/Water Library

Properties are calculated via the Equations of State which were used to generate the classic steam/water tables in the IAPWS Industrial Formulation 1997 for the Thermodynamic Properties of Water and Steam ([ISPWS-IF97](#)), see [ASME Press in References](#).

After selecting ASME Steam/Water, "ASME '97 Water" is automatically added to in the Fluids in Current Model area. Select the Fluid Phase and enter a Pressure and Temperature to calculate the properties as shown in Figure 2.

Detailed fluid property information from SteamCalc can be calculated using the AFT SteamCalc Viewer, which can be accessed from Tools > Fluid Property Viewers > SteamCalc. The SteamCalc Viewer will be launched as a separate application, which can then be used to calculate fluid properties at various conditions.

For more information on AFT SteamCalc Viewer see the [SteamCalc Help File](#).

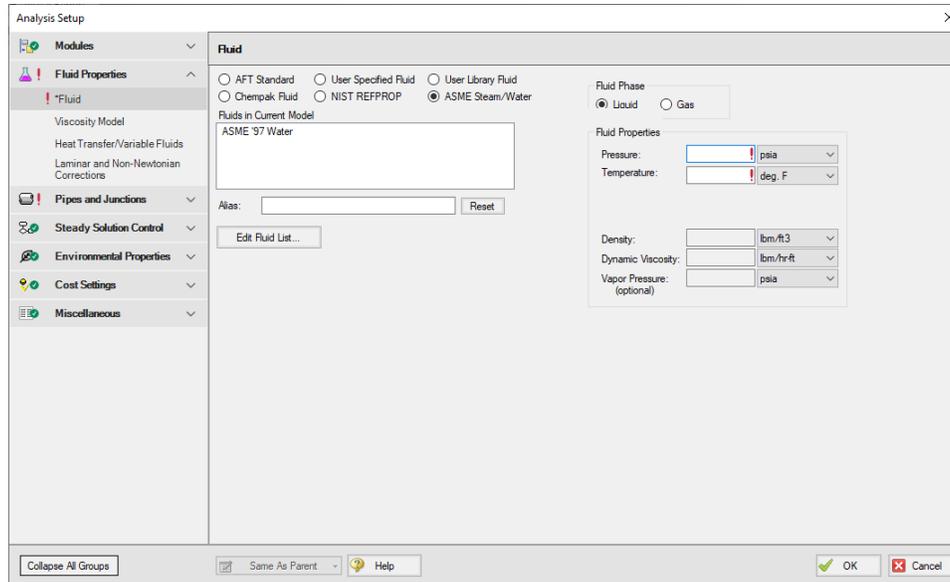


Figure 2: Fluid panel in Analysis Setup with the ASME Steam/Water fluid library selected

NIST REFPROP Library

The Reference Fluid Thermodynamic and Transport Properties (REFPROP) Database is licensed from the National Institute of Standards and Technology (NIST) where properties for approximately 150 fluids are calculated via Equations of State. The data comes from Version 10 of the library. For more info see the following sources: [Program Summary](#), [REFPROP FAQ](#), [NIST Chemistry WebBook](#), [NIST Chemical Name Search](#).

Add a fluid from the Fluids Available in Library list by selecting it and clicking Add to Model. Select the Fluid Phase and enter both a Pressure and Temperature in the Fluid Properties area on the right, as shown in Figure 3.

Fluids from the NIST REFPROP library support [fluid mixtures](#), but no chemical reactions are modeled as listed in the [Engineering Assumptions](#).

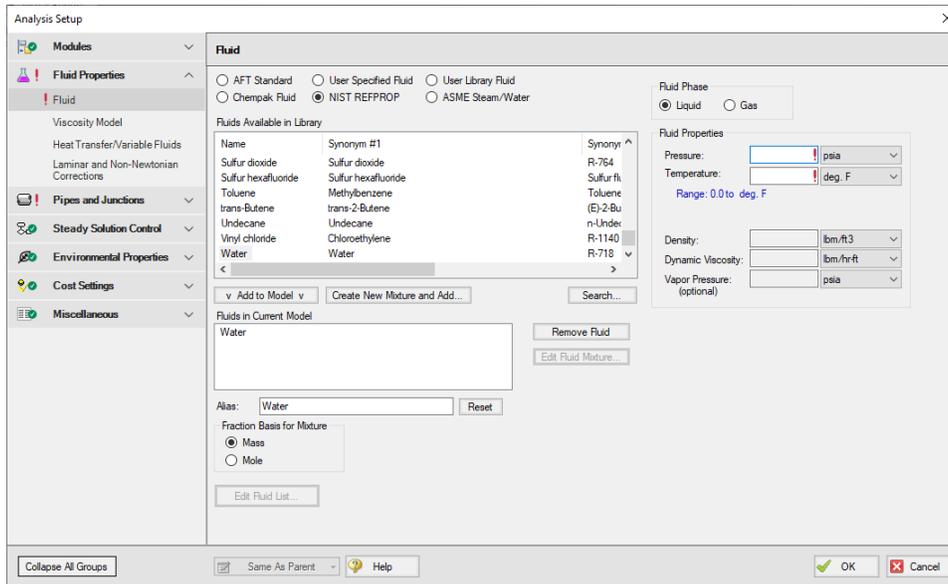


Figure 3: Fluid panel in Analysis Setup with the NIST REFPROP fluid library selected

Chempak Fluid Library

Optional add-on for purchase which contains data for approximately 700 fluids. Chempak is licensed from Madison Technical Software.

Fluids from the Chempak library support [fluid mixtures](#), but no chemical reactions are modeled as listed in the [Engineering Assumptions](#).

Add a fluid from the Fluids Available in Library list by selecting it and clicking Add to Model. Select the Fluid Phase and enter both a Pressure and Temperature in the Fluid Properties area on the right, as shown in Figure 4.

Information on the theory and calculation methods used in Chempak is given in the [Chempak Technical Information](#) topic.

Detailed fluid property information from Chempak can be calculated using the AFT Chempak Viewer, which can be accessed from Tools > Fluid Property Viewers > Chempak. The Chempak Viewer will be launched as a separate application, which can then be used to calculate fluid properties at various conditions. A Chempak Database license is required to use the Chempak Viewer.

For more information on AFT Chempak Viewer see the [Chempak Viewer Help File](#).

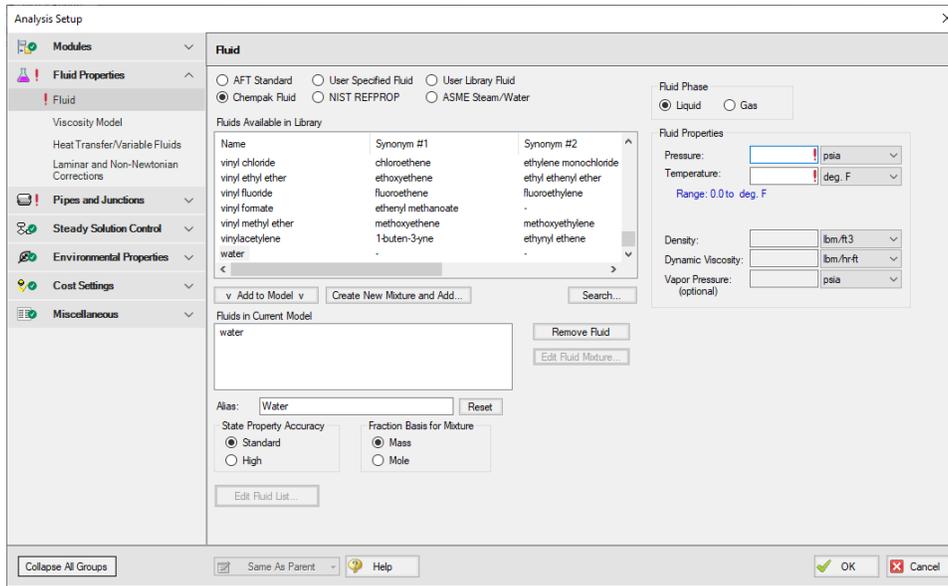


Figure 4: Fluid panel in Analysis Setup with the Chempak fluid library selected

State Property Accuracy Options

The State Property Accuracy options are applicable only to Chempak fluids and are located on the Fluid panel below the Fluids in Current Model area.

The State Property Accuracy influences the number of elements used in curve fits for calculating property data. There are two accuracy options: Standard and High. You will usually want to start with standard accuracy, which is selected by default, and use high accuracy for final runs, depending on the importance and run time for the model. The high accuracy option will increase run time by about a factor of two.

When performing interpolations for properties, a 4 point or 16 point interpolation scheme can be used. The 4 point scheme implemented in AFT Fathom is the Standard Accuracy option. The 16 point scheme is the high accuracy option. Use of the High accuracy option will roughly double the run times for models.

Related Topics

Fluid Properties Group.....	90
Viscosity Model Panel.....	97
Heat Transfer/Variable Fluids Panel.....	120
Laminar and Non-Newtonian Corrections Panel.....	123
SSL Module Fluid Properties Group.....	131
Creating Liquid Mixtures.....	96
Transport Properties.....	97
Chempak temperature and or pressure is out of range for the following.....	776
Static pressure lower than vapor pressure at inlet/outlet of pipe/junction X.....	782

Temperature at inlet/exit of pipe X is above/below max/min fluid temperature range..... 782
 Temperature is above the saturation line for the following.....783

Creating Liquid Mixtures

You have the option of creating a mixture of fluids with the NIST REFPROP or Chempak fluid libraries.

To create a mixture, first select NIST REFPROP or Chempak Fluid on the [Fluid panel](#) in Analysis Setup. Once this has been selected, the option to Create New Mixture and Add button becomes available. Click this button, then locate and select the first fluid in the mixture you wish to specify (the Search button can be used to search for a name or synonym). Enter the mass or mole fraction of the fluid in the mixture, and click Add Fluid to Mixture. Repeat this process for all fluids in the mixture as shown in Figure 1.

Note: The fraction basis of all mixtures is either mass or mole, and it is specified on the Fluid panel itself. If you would like to change the fraction basis for the mixture (from mass to mole or from mole to mass), click on the desired fraction basis located at the bottom of the Fluid panel below Fraction Basis for Mixture. Note that this option is only available after you specify the model fluid as a mixture.

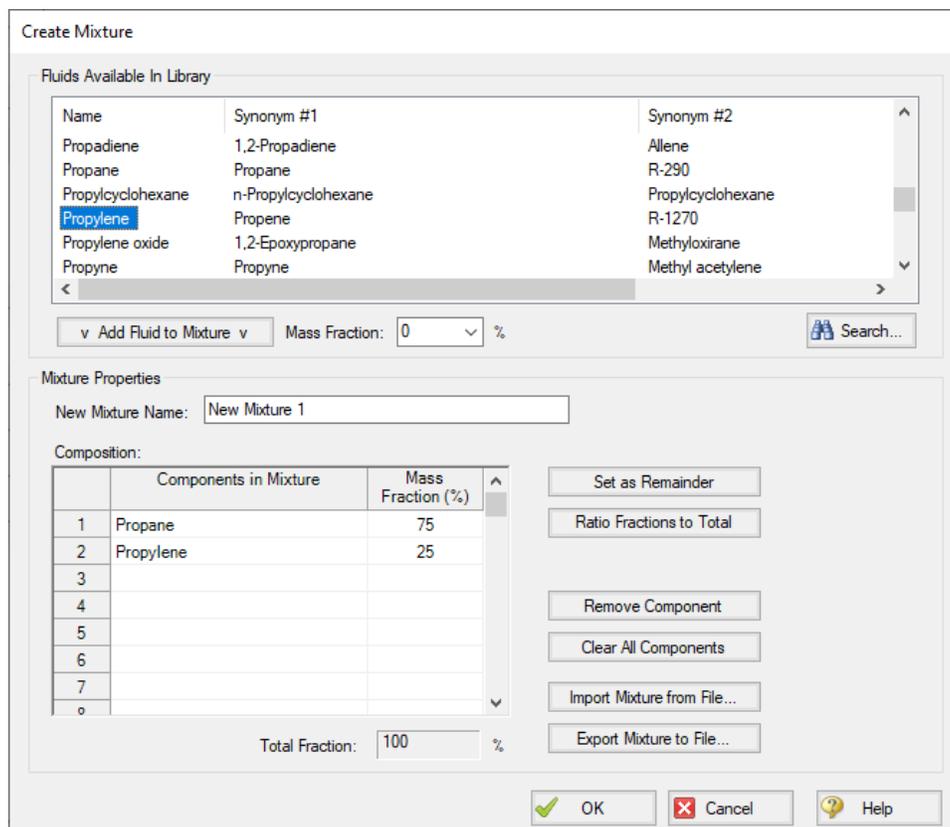


Figure 1: The Create Mixture window (opened from the Fluid panel) allows you to create predefined mixtures for the model

To add the mixture to the Fluid panel, the Total Fraction for the table must equal 100%. To help achieve this, the Set as Remainder button can be used to add the remaining percentage to the selected

component in the table so that the fractions in the table will add up to 100%. Alternatively, the Ratio Fractions to Total button will scale up the fractions in the table so that they will add to 100%.

Components can be deleted from the table by using the Remove Component or Clear All Components buttons.

Mixture data can be exported to a file which can then be imported into other AFT applications, or shared with other AFT application users by using the Export Mixture to File and Import Mixture from File buttons in this window.

Note: The NIST REFPROP fluid library is not able to calculate the mixing parameters for most mixtures containing water, helium, hydrogen, neon, carbon monoxide, fluorine, methanol, or ethanol.

Related Topics

Fluid Panel.....	91
Transport Properties.....	97

Transport Properties

The transport properties in AFT Fathom (specific heat, dynamic viscosity, and thermal conductivity) are assumed to be functions of temperature only. In general this is a good assumption, although systems at very high pressure usually start to show some pressure dependence. These properties are modeled as polynomial curve fits of temperature or interpolated.

Related Topics

Fluid Panel.....	91
Creating Liquid Mixtures.....	96

Viscosity Model Panel

AFT Fathom can model systems with Newtonian or non-Newtonian fluids. A Newtonian fluid is one where the fluid viscosity is fixed and independent of fluid velocity, while a non-Newtonian fluid is one where the fluid viscosity is variable and dependent on fluid velocity. Most industrial fluids are Newtonian fluids, although there are important applications for non-Newtonian fluids. By default, AFT Fathom has the Newtonian viscosity model selected.

AFT Fathom offers several viscosity models:

- Newtonian (default)
- [Paper Stock - Duffy](#)
- [Paper Stock - Brecht & Heller](#)
- [Power Law](#)
- [Bingham Plastic](#)

- [Herschel-Bulkley](#)
- [Homogeneous Scale-up](#)

Modeling non-Newtonian fluids is a complex task. A myriad of theoretical models and test equipment exist, and each have various names unique to an industry or application. Each of the non-Newtonian viscosity models require rheological inputs to be defined. Those inputs are detailed on the pages for each model as well as an explanation as to what the viscosity model is meant for, how to use the model, and how the model calculates hydraulic behavior in a system.

Frictional Losses

Each non-Newtonian fluid model adjusts the frictional losses in the system from the Newtonian fluid model. The equations and correlations used to determine pipe frictional losses are detailed on each model's page. These methods to determine frictional losses replace the pipe friction model specified in the Pipe Properties window. For pipe Fittings and junction-specific losses, all non-Newtonian models use the same method, as detailed in the [Laminar and Non-Newtonian Corrections panel](#) topic.

Related Topics

Fluid Properties Group.....	90
Fluid Panel.....	91
Heat Transfer/Variable Fluids Panel.....	120
Laminar and Non-Newtonian Corrections Panel.....	123
SSL Module Fluid Properties Group.....	131
Duffy Method (Pulp and Paper Stock).....	98
Brecht & Heller Method (Pulp and Paper Stock).....	101
Power Law Model.....	103
Bingham Plastic Model.....	106
Homogeneous Scale-up Method.....	112
Herschel-Bulkley Model.....	108
Using Rheological Data.....	115

Related Examples

Non-Newtonian Phosphates Pumping.....	918
Non-Newtonian Mill Discharge Slurry.....	974

Duffy Method (Pulp and Paper Stock)

The Duffy method is a system developed by G.G. Duffy at the University of Auckland, New Zealand and formalized by the [Technical Association of the Pulp and Paper Industry \(TAPPI\)](#). The model uses correlations based on data obtained from laboratory flow loops and real-world pulp mill piping systems.

TAPPI states that the correlations behind this model do not necessarily provide the actual design value of friction loss. Instead, it provides a means to determine the minimum value of pipe friction loss an

engineer can expect. Each system designer must determine and apply a safety factor as they deem necessary.

Fathom contains data on 21 different stock fluid selections that can be used with the Duffy method. The constants used for each stock fluid are displayed in the General Results section of the [output](#). Additionally, the Vmax and Vwater terms, the dryness coefficient, and the Safety Factor can all be displayed in the Pipe Output.

Note: The Duffy method is only valid for pulp consistencies between 2% and 6%.

Duffy Method Interface

The Viscosity Model panel appears as shown below in Figure 1 when the Duffy method is selected.

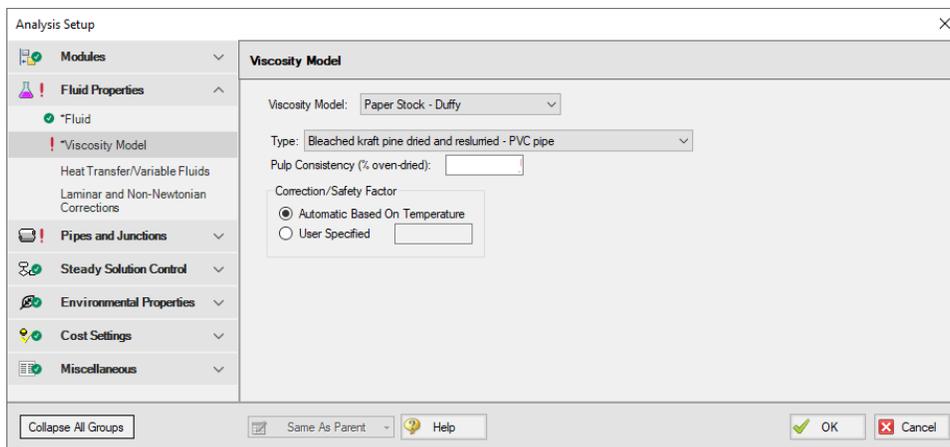


Figure 1: Viscosity Model panel for the Duffy Method

Users must select a type of pulp to model and define a pulp consistency. The user must also choose to correct the Duffy method parameters based on temperature automatically or via a user-specified correction. The constants used by Fathom are based on a pulp temperature of 35 deg. C. The Automatic Based on Temperature will use the temperature specified on the Fluid panel (or the temperature in each pipe in the case of [Variable Fluid Properties](#)) to determine the pulp temperature.

There are 21 pulp types to choose from:

- Bleached kraft pine dried and reslurried - PVC pipe
- Bleached kraft pine dried and reslurried - SS pipe
- Bleached straw - copper pipe
- Bleached sulfite - copper pipe
- Hardwood NSSC CSF = 620 - PVC pipe
- Kraft - copper pipe
- Kraft birch dried and reslurried - PVC pipe
- Long-fibered kraft dried and reslurried - PVC pipe

- Long-fibered kraft never dried CSF = 260 - PVC pipe
- Long-fibered kraft never dried CSF = 550 - PVC pipe
- Long-fibered kraft never dried CSF = 650 - PVC pipe
- Long-fibered kraft never dried CSF = 725 - PVC pipe
- Long-fibered kraft never dried CSF = 725 - SS pipe
- Newsprint broke CSF = 75 - PVC pipe
- Refiner groundwood CSF = 150 - PVC pipe
- Refiner groundwood (hardboard) - PVC pipe
- Refiner groundwood (insulating board) - PVC pipe
- Stone groundwood CSF = 114 - PVC pipe
- Unbeaten aspen sulfite never dried - SS pipe
- Unbleached straw - copper pipe
- Unbleached sulfite - copper pipe

Theory and Correlations

AFT Fathom models frictional losses for the Duffy method using correlations taken from the TAPPI Generalized Method for Determining the Pipe Friction Loss of Flowing Pulp Suspensions.

The maximum velocity (m/s) for the Duffy model is given by:

$$V_{max} = K' C^{\sigma}$$

The velocity at which pulp effects are negligible and the frictional losses of the pulp mixture are the same as the frictional losses of pure water:

$$V_{water} = 1.22 C^{1.40}$$

In the case where $V < V_{max}$:

$$\frac{dH}{L} = F K V^{\alpha} C^{\beta} D^{\gamma}$$

Where:

- dH/L = head loss per length (m water/100 m pipe)
- C = pulp consistency (% dryness)
- D = inner pipe diameter (mm)
- F = correction factor for temperature or pipe roughness (dimensionless)
- $K', K, \sigma, \alpha, \beta, \gamma$ = constants depending on pulp type and pipe material (dimensionless)
- V = actual pipe velocity, when $V_{max} > V$ (m/s)
- $V = V_{max}$, when $V_{water} > V > V_{max}$

When pipe velocity is larger than V_{water} , [standard pipe friction models](#) are used, treating the pulp mixture as pure water.

Related Topics

Viscosity Model Panel.....	97
Brecht & Heller Method (Pulp and Paper Stock).....	101
Power Law Model.....	103
Bingham Plastic Model.....	106
Homogeneous Scale-up Method.....	112
Herschel-Bulkley Model.....	108

Brecht & Heller Method (Pulp and Paper Stock)

The Brecht and Heller method is a system developed by Brecht and Heller at the University of Maine and formalized by the Technical Association of the Pulp and Paper Industry (TAPPI) and the [Cameron Hydraulic Handbook](#). The model uses correlations based on data obtained from laboratory flow loops and real-world pulp mill piping systems.

This method is considered more conservative than the [Duffy method](#), meaning the Brecht and Heller method will predict larger pressure losses in a system, all else constant.

Fathom contains data on eight different stock fluid selections that can be used with the Brecht and Heller method. The constants used for each stock fluid are displayed in the General Results section of the [output](#). Additionally, the pseudo-Reynolds Number ($Ré$) and Friction Factor terms can be displayed in the Pipe Output.

Brecht and Heller Method Interface

The [Viscosity Model panel](#) appears as shown below in Figure 1 when the Brecht & Heller method is selected.

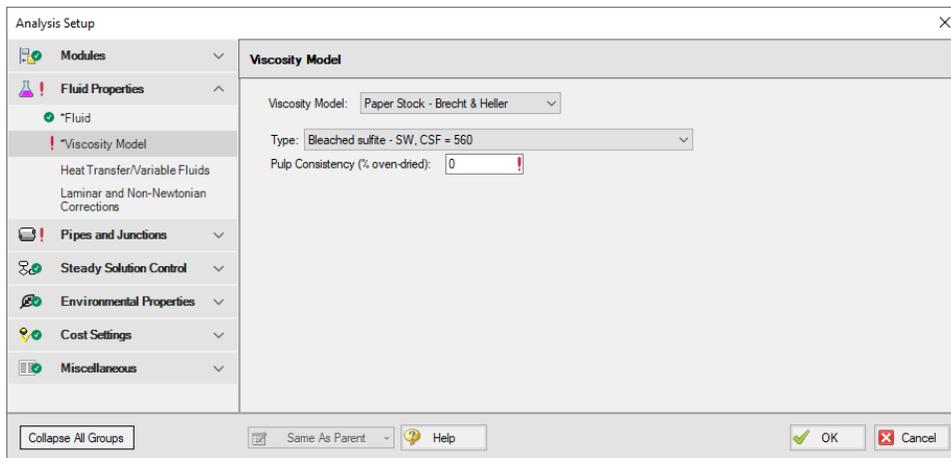


Figure 1: Viscosity Model panel for the Brecht & Heller Method

Users must select a type of pulp model and define a pulp consistency. There eight pulp types to choose from:

- Bleached sulfite - SW, CSF = 560
- Pre-steamed groundwood - SW, CSF = 200
- Reclaimed fiber
- Soda - HW
- Stone groundwood - SW, CSF = 70
- Unbleached kraft - SW, CSF 730
- Unbleached sulfite - SW, CSF = 640
- Unbleached, unrefined sulfite SW

Theory and Correlations

AFT Fathom models frictional losses for the Brecht and Heller method using correlations taken from the Cameron Hydraulic Handbook, pg 3-88.

The pseudo-Reynolds number for the paper stock (Re') is:

$$Re' = \frac{\rho V D^{0.205}}{C^{1.157}}$$

The pipe friction factor is:

$$f = \frac{3.97}{Re'^{1.636}}$$

The friction loss in feet along the pipe is:

$$dH = \frac{fV^2 L K}{D}$$

Where:

- D = inner diameter (feet)
- L = pipe length (feet)
- V = average velocity (feet/sec)
- ρ = stock fluid density (lbm/ft³)
- C = pulp consistency (% dryness)
- K = friction factor constant, specific to pulp type (dimensionless)

Related Topics

Viscosity Model Panel.....	97
Duffy Method (Pulp and Paper Stock).....	98
Power Law Model.....	103
Bingham Plastic Model.....	106
Homogeneous Scale-up Method.....	112
Herschel-Bulkley Model.....	108

Power Law Model

The Power Law model for non-Newtonian fluids is one of the most common viscosity models used in hydraulic analysis. This model states that the shear stress required to shear a fluid at a given rate can be described by the following equation:

$$\tau = K \left(\frac{du}{dy} \right)^n$$

Where τ is the shear stress, du/dy is the shear rate (or velocity gradient), and K and n are the Power Law constants.

The Power Law model defines η , the apparent viscosity of the fluid as:

$$\eta = K \left(\frac{du}{dy} \right)^{n-1}$$

In the Power Law model, a value of $n=1$ corresponds to a Newtonian fluid, while $0 < n < 1$ is a Shear Thinning (pseudoplastic) fluid and $n > 1$ represents a Shear Thickening (dilatant) fluid.

To employ this model, users must have values for K and n , or have [rheological test data](#).

Power Law Interface

The Viscosity Model panel appears as shown below in Figure 1 when the Power Law model is selected.

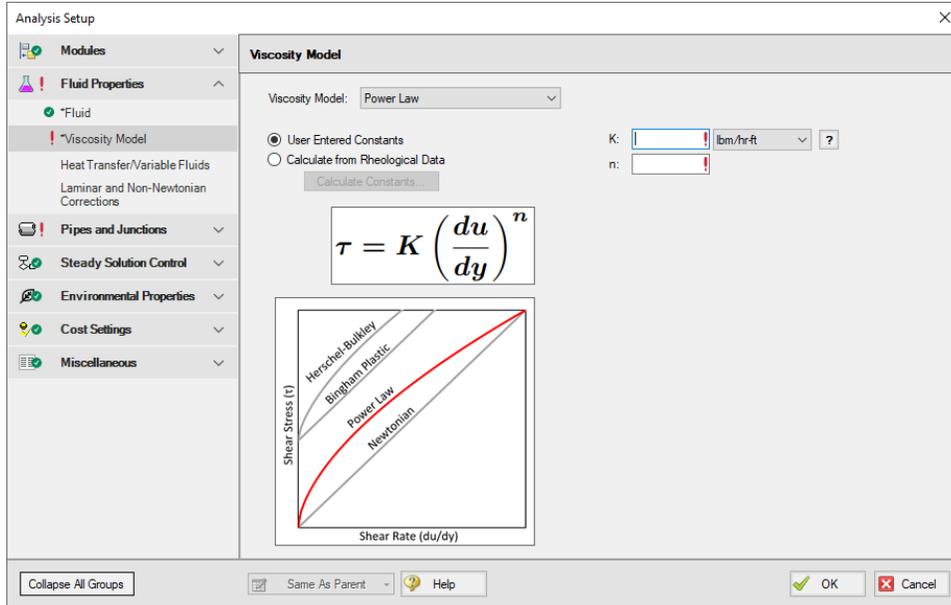


Figure 1: Viscosity Model panel for the Power Law model

Users must enter values for the Power Law constants, K and n . If the user has raw rheological test data instead, they can enter that directly using the [Calculate from Rheological Data](#) option.

Theory and Correlations

AFT Fathom models the frictional losses due to a Power Law fluid using correlations taken from [Darby](#), pg. 166-167. Note that the original equations in Darby use the Fanning friction factor, while AFT software uses the Moody friction factor. The equations below have been modified from the original accordingly.

$$f = (1 - \alpha) f_L + \frac{\alpha}{(f_T^{-8} + F_{Tr}^{-8})^{\frac{1}{8}}}$$

Where:

$$f_L = \frac{64}{Re_{pl}}$$

$$f_T = \frac{0.2728 n^{-0.5}}{Re_{pl}^{1/(1.87 + 2.39n)}}$$

$$f_{Tr} = 7.16 * 10^{-4} e^{-5.24n} Re_{pl}^{0.414 + 0.757n}$$

$$\alpha = \frac{1}{1 + 4^{-\Delta}}$$

$$\Delta = Re_{pl} - Re_{plc}$$

The Reynolds number for power law fluid (Re_{pl}) is defined as:

$$Re_{pl} = \frac{8 \rho V^{2-n} D^n}{K \left(\frac{2(3n+1)}{n} \right)^n}$$

The critical Reynolds number for power law fluid (Re_{plc}) where laminar flow ceases is defined as:

$$Re_{plc} = 2100 + 875 (1 - n)$$

The friction factor calculated using this correlation is then used to determine pressure loss following [Equation 3](#) in the [Network Solution Methodology](#) topic.

These correlations are valid across the full range of Reynolds number. The applicability of this method to highly turbulent flow is limited only by the test data used to determine the Power Law constants.

Related Topics

Viscosity Model Panel.....	97
Duffy Method (Pulp and Paper Stock).....	98
Brecht & Heller Method (Pulp and Paper Stock).....	101
Bingham Plastic Model.....	106
Homogeneous Scale-up Method.....	112
Herschel-Bulkley Model.....	108
Using Rheological Data.....	115

Related Examples

Non-Newtonian Phosphates Pumping.....	918
Non-Newtonian Mill Discharge Slurry.....	974

Bingham Plastic Model

The Bingham Plastic model for non-Newtonian fluids is one of the most common viscosity models used in hydraulic analysis. This model states that the shear rate resulting from a given shear stress applied to the fluid can be described with the following piecewise equation:

$$\frac{du}{dy} = \begin{cases} 0, & \tau < \tau_o \\ \frac{(\tau - \tau_o)}{\mu_\infty}, & \tau \geq \tau_o \end{cases}$$

Where du/dy is the shear rate (or velocity gradient), τ is the applied shear stress, τ_o is the yield stress, and μ_∞ is the plastic viscosity or coefficient of rigidity.

The Bingham Plastic model describes η , the apparent viscosity of the fluid as:

$$\eta = \frac{\tau_o}{du/dy} + \mu_\infty$$

The Bingham Plastic model uses a constant value for plastic viscosity, meaning a value for μ_∞ of 0 would result in a Newtonian fluid. Some fluids may see Bingham Pseudoplastic behavior, where shear stress and shear rate do not have a linear relationship. These fluids can be modeled using the [Herschel-Bulkley viscosity model](#).

To employ this model, users must have values for τ_o and μ_∞ , or have [rheological test data](#).

Bingham Plastic Interface

The [Viscosity Model panel](#) appears as shown below in Figure 1 when the Bingham Plastic model is selected.

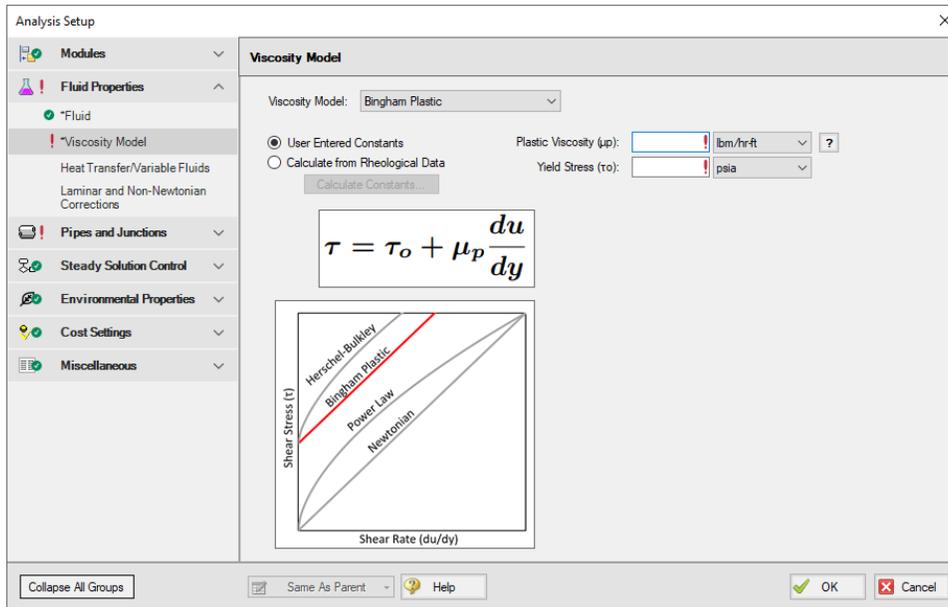


Figure 1: Viscosity Model panel for the Bingham Plastic model

Users must enter values for the Bingham Plastic constants, μ_{∞} and τ_o . If the user has raw rheological test data instead, they can enter that directly using the [Calculate from Rheological Data](#) option instead.

Theory and Correlations

AFT Fathom models the frictional losses due to a Bingham Plastic fluid using correlations taken from [Darby](#), pg. 168-169. Note that the original equations in Darby use the Fanning friction factor, while AFT software uses the Moody friction factor. The equations below have been modified from the original accordingly.

$$f = (f_L^m + f_T^m)^{\frac{1}{m}}$$

Where:

$$m = 1.7 + \frac{40,000}{Re}$$

$$f_L = \frac{64}{Re} \left(1 + \frac{He}{6 Re} - \frac{64He^4}{3f_L^3 Re^7} \right)$$

$$f_T = 4 \left(\frac{10^a}{Re^{0.193}} \right)$$

$$a = -1.47 \left(1 + 0.146e^{-2.9 \times 10^{-5} He} \right)$$

The Reynolds number for a Bingham Plastic fluid is defined as:

$$Re = \frac{\rho V D}{\mu_\infty}$$

The Hedstrom number for a Bingham Plastic fluid is defined as:

$$He = \frac{\rho \tau_o D^2}{\mu_\infty^2}$$

The friction factor calculated using this correlation is then used to determine pressure loss following [Equation 3](#) in the [Network Solution Methodology](#) topic.

These correlations are valid across the full range of Reynolds number. The applicability of this method to highly turbulent flow is limited only by the test data used to determine the Bingham Plastic constants.

Related Topics

Viscosity Model Panel.....	97
Duffy Method (Pulp and Paper Stock).....	98
Brecht & Heller Method (Pulp and Paper Stock).....	101
Power Law Model.....	103
Homogeneous Scale-up Method.....	112
Herschel-Bulkley Model.....	108
Using Rheological Data.....	115

Related Examples

Non-Newtonian Phosphates Pumping.....	918
Non-Newtonian Mill Discharge Slurry.....	974

Herschel-Bulkley Model

The Herschel-Bulkley model for non-Newtonian fluids states that the shear rate resulting from a given shear stress applied to the fluid can be described with the following equation:

$$\tau = \tau_o + K \left(\frac{du}{dy} \right)^n$$

Where du/dy is the shear rate (or velocity gradient), τ is the applied shear stress, τ_o is the yield stress, K is the consistency coefficient, and n is the flow behavior index.

The Herschel-Bulkley model describes describes η , the apparent viscosity of the fluid as:

$$\eta = \frac{\tau_o}{du/dy} + K \left(\frac{du}{dy} \right)^{n-1}$$

In the Herschel-Bulkley model a value for τ_o of 0 and a value for n of 1 would result in a Newtonian fluid. The Herschel-Bulkley model is equivalent to the Bingham Plastic model if $n = 1$, and is equivalent to the Power Law model if $\tau_o = 0$.

To employ this model, users must have values for τ_o , K and n , or have [rheological data](#).

Herschel-Bulkley Interface

The Viscosity Model panel appears as shown below in Figure 1 when the Herschel-Bulkley model is selected.

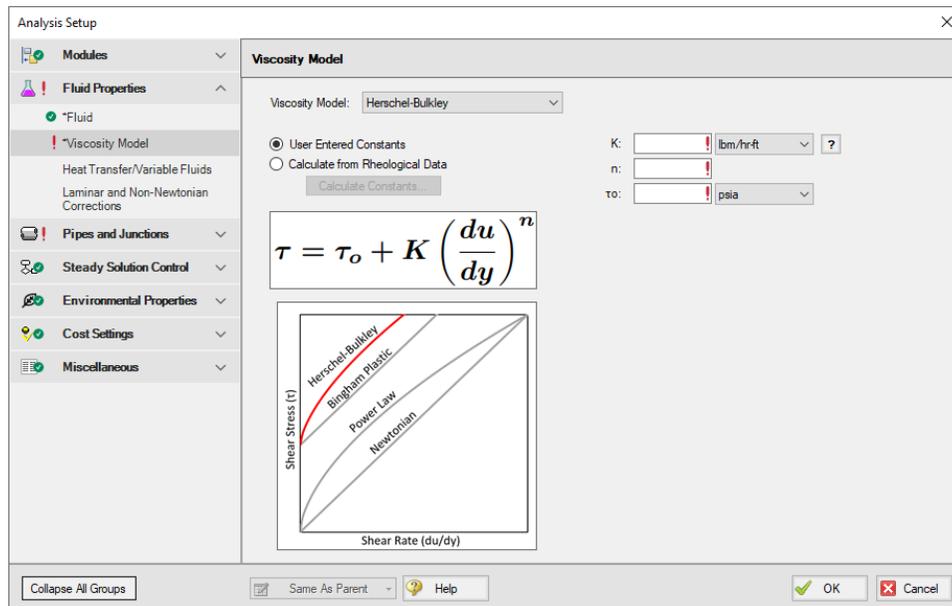


Figure 1: Viscosity Model panel for the Herschel-Bulkley model

Users must enter values for the Herschel-Bulkley constants, K , n , and τ_o . If the user has raw rheological test data instead, they can enter that directly using the [Calculate from Rheological Data](#) option.

Theory and Correlations

AFT Fathom models the frictional losses due to a Herschel-Bulkley fluid using correlations as are summarized in [Garcia & Steffe](#) (1986). Note that the original equations in the reference use the Fanning friction factor, while AFT software uses the Moody friction factor.

The **laminar flow** friction factor is defined as

$$f_{L,fanning} = \frac{16}{\psi Re}$$

with

$$\psi = (1 + 3n)^n (1 - \xi_o)^{1+n} \left[\frac{(1-\xi_o)^2}{(1+3n)} + \frac{2\xi_o(1-\xi_o)}{(1+2n)} + \frac{\xi_o^2}{(1+n)} \right]^n$$

and ξ_o , the dimensionless unsheared plug radius:

$$\xi_o = \frac{\tau_o}{\tau_w} = \frac{\tau_o}{0.5 f \rho v^2}$$

The **turbulent flow** friction factor can be calculated as follows

$$f_{T,fanning} = \left[0.45 - \frac{2.75}{n} + \frac{1.97}{n} \ln(1 - \xi_o) + \frac{1.97}{n} \ln \left(Re \left(\frac{1+3n}{4n} \right)^n f^{1-(n/2)} \right) \right]^{-2}$$

where

$$\xi_o = \frac{16(2 He)^{n/(2-n)} \left(\frac{n}{1+3n} \right)^{2n/(2-n)}}{Re^{2/(2-n)} f}$$

Where He is the Hedstrom number for Herschel-Bulkley fluids and is defined as

$$He = \frac{D^2 \rho}{K} \left(\frac{\tau_o}{K} \right)^{\frac{2-n}{n}}$$

Re is the Reynolds number for Herschel-Bulkley fluids, and is equivalent to the Reynolds number for Power Law fluids:

$$Re_{pl} = \frac{8 \rho V^{2-n} D^n}{K \left(\frac{2(3n+1)}{n} \right)^n}$$

The **critical Reynolds number**, which represents the transition from laminar to turbulent flow for Herschel Bulkley fluids is defined as

$$Re_c = \frac{6464 n \psi_c^{(2/n)-1} (2+n)^{(2+n)/(1+n)}}{(1+3n)^2 (1-\xi_{oc})^{(2/n)+1}}$$

Where n is the flow behavior index, ψ_c is equivalent to Ψ with $\xi_o = \xi_{oc}$, and ξ_{oc} is defined implicitly as follows

$$He = \frac{3232(2+n)^{(2+n)/(1+n)} \xi_{oc}^{(2/n)-1}}{n(1-\xi_{oc})^{(2/n)+1}}$$

The friction factor calculated using this correlation is then used to determine pressure loss following [Equation 3](#) in the [Network Solution Methodology](#) topic.

These correlations are valid across the full range of Reynolds number. The applicability of this method to highly turbulent flow is limited only by the test data used to determine the Herschel-Bulkley constants.

Curve Fit Methodology

Unlike the Power Law and Bingham Plastic models the Herschel-Bulkley method uses three parameters, so a simple linear regression cannot be used to curve fit all three constants.

Instead Fathom employs the method discussed in [Mullineux \(2008\)](#), which first solves a root-finding equation to determine n , then performs a non-linear regression of τ vs x^n to determine the constants K and τ_o .

The curve fit results using this method have been proven to be identical to the results of a non-linear least squares regression, which is one of the most common methods used to curve fit the Herschel-Bulkley parameters.

If a different curve fitting method is preferable to represent the fluid being modeled the User Entered Constants option should be selected instead.

Related Topics

Viscosity Model Panel.....	97
Duffy Method (Pulp and Paper Stock).....	98
Brecht & Heller Method (Pulp and Paper Stock).....	101
Power Law Model.....	103
Bingham Plastic Model.....	106

Homogeneous Scale-up Method..... 112
 Using Rheological Data..... 115

Homogeneous Scale-up Method

The Homogeneous Scale-up method for non-Newtonian fluids is based on the idea that test data for velocity and pressure drop along pipes of a given diameter can be scaled up to model the velocity and pressure drop for a pipe of a large size.

This scale-up is similar to using the [affinity laws](#) to predict pump behavior. Instead of scaling pressure rise and volumetric flowrate for a pump, the Homogeneous Scale-up method scales the frictional gradient (j_m) and mean velocity (V_m) for a pipe.

To employ this method, users must have test data from the same [Tube Flow Rheometer](#) as is available when entering rheological data for the Power law, Bingham Plastic, and Herschel-Bulkley models.

Homogeneous Scale-up Interface

The Viscosity Model panel appears as shown below in Figure 1 when the Homogeneous Scale-up method is selected. Click the Enter Scaled Data button to enter data.

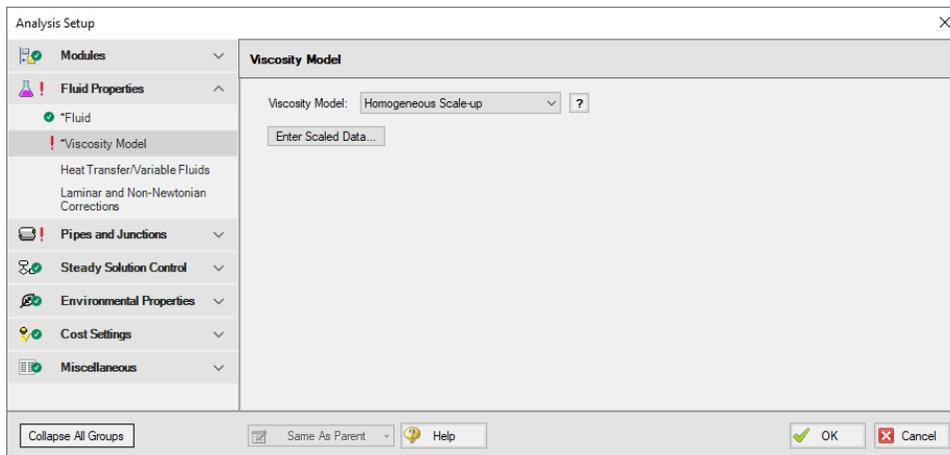


Figure 1: Viscosity Model panel for the Homogeneous Scale-up method

The Homogeneous Scale-up window is shown below in Figure 2. Here, the user must specify information about the pipe used to obtain the test data along with the test data itself. The flow data can be entered as mass flowrate, velocity, volumetric flowrate, or shear rate data as determined by a tube flow rheometer. The Head/Pressure data can be entered in terms of Head gradient, Pressure gradient, or shear stress data as determined by a tube flow rheometer.

Each data point must also be marked as laminar or turbulent flow.

Homogeneous Scale-up

Raw Data Graph

Test Pipe Material Data

Material: Steel - ANSI

Size: 8 inch

Type: STD (schedule 40)

Inner Diameter: 7.981 inches

	Flow Velocity (feet/sec)	Head/Pressure Jm (ft/100 ft)	Flow Regime
Parameter	Velocity	Jm	
Units	feet/sec	ft/100 ft	
1	1.74	0.089	Laminar
2	4.99	0.1	Laminar
3	6.56	0.102	Laminar
4	8.5	0.105	Laminar
5	10.6	0.108	Laminar
6	12.5	0.11	Laminar
7	14.5	0.113	Laminar
8	16.8	0.119	Turbulent
9	18.5	0.13	Turbulent
10			Turbulent
11			Turbulent
12			Turbulent
13			Turbulent

Edit Table Make All Laminar All Turbulent After First Turbulent Create Graph

OK Cancel Help

Figure 2: Homogeneous Scale-up Data entry window

The user can also generate a graph of their data plotted against other viscosity models and larger scaled pipe sizes. This interface is shown in Figure 3 below. Note that generating a cross-plot for the Power Law or Bingham Plastic models does not mean Fathom will use those models in the system. The cross-plot is intended only to give a visual comparison of the data.

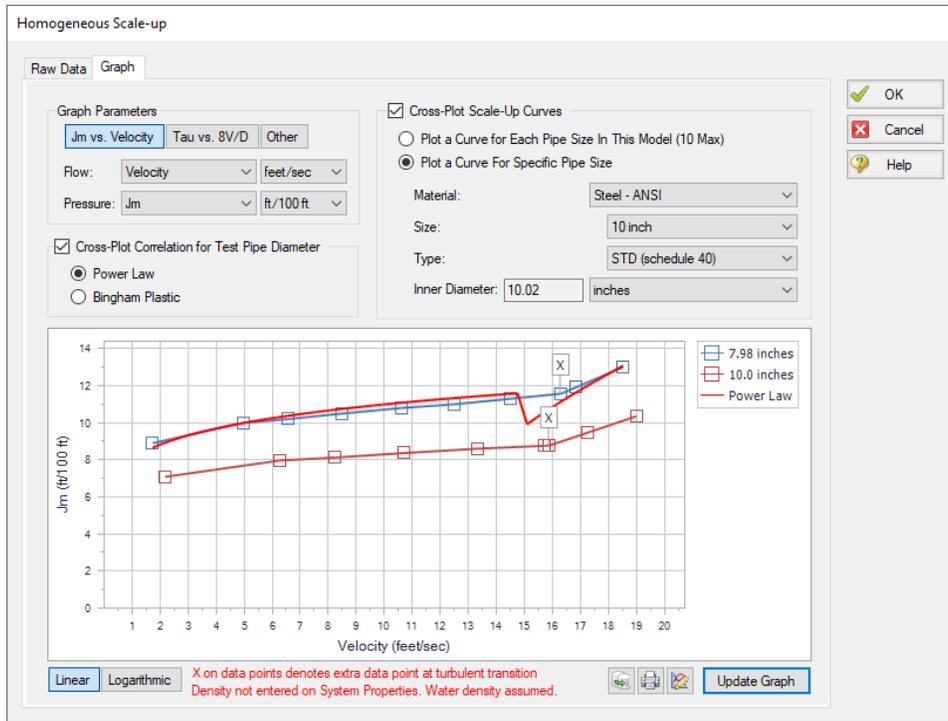


Figure 3: Graphed data for the Homogeneous Scale-up method

Theory and Correlations

The Homogeneous Scale-up methods are based on Chapter 3 of [Wilson, et al, 2006](#), page 67.

Laminar data scales as follows:

$$(j_m)_2 = (j_m)_1 + \frac{D_1}{D_2}$$

$$(V_m)_2 = (V_m)_1 + \frac{D_2}{D_1}$$

Turbulent data scales as follows:

$$(j_m)_2 = (j_m)_1 + \frac{D_1}{D_2}$$

$$(V_m)_2 = (V_m)_1 + 2.5 U \ln \left(\frac{D_2}{D_1} \right)$$

$$U = \sqrt{\frac{\tau}{\rho}}$$

Where:

- Data point 1 is the test data
- Data point 2 is the larger system
- τ is the shear stress at the pipe wall

The pressure drop calculated using these methods is then substituted into the network solution and iterated upon until a converged solution is available.

These correlations are valid across the full range of Reynolds number. The applicability of this method to highly turbulent flow is limited only by the test data the user has available.

Related Topics

Viscosity Model Panel.....	97
Duffy Method (Pulp and Paper Stock).....	98
Brecht & Heller Method (Pulp and Paper Stock).....	101
Power Law Model.....	103
Bingham Plastic Model.....	106
Herschel-Bulkley Model.....	108
Using Rheological Data.....	115
Affinity Laws.....	324

Related Examples

Non-Newtonian Phosphates Pumping.....	918
Non-Newtonian Mill Discharge Slurry.....	974

Using Rheological Data

For the [Power Law](#), [Bingham Plastic](#) and [Herschel-Bulkley](#) viscosity models, users are expected to define constants for each model. If the user already has these constants, they can be entered directly into the software. If the user only has the original rheological data, Fathom can calculate the viscosity model constants from that data.

The interface used to enter this data and the processes Fathom employs to manipulate it are similar for all of the viscosity models mentioned above. This page discusses rheological data only in the context of the Power Law model for simplicity. Unless specified otherwise, the content discussed here is applicable to all models.

Calculate from Rheological Data Interface

The Calculate from Rheological Data option can be accessed from the Viscosity Model panel in Analysis Setup. Click the Calculate from Rheological Data radio button and click the Calculate Constants button as shown below in Figure 1.

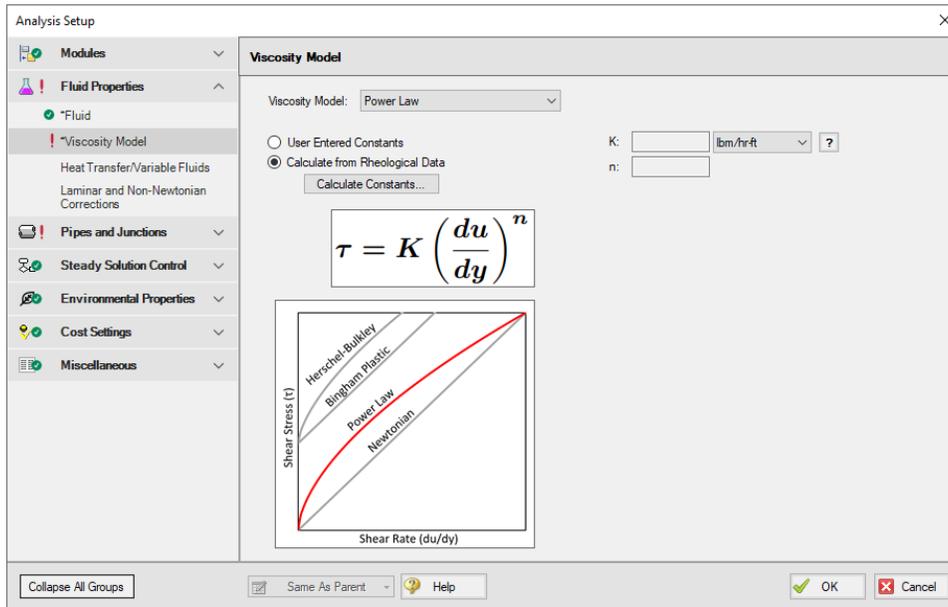


Figure 1: Power Law viscosity model using the Calculate from Rheological Data option

The Calculate Power Law Constants window will open to enter rheological data. This window is shown below in Figure 2. In this window, users are asked to enter Shear Stress (τ) vs. Shear Rate data in units of pressure and inverse time, respectively.

Two Raw Data Type options are presented above the table, Tube Flow Rheometer Data ($8V/D$) and Shear Rheometer Data (du/dy). These options refer to the experimental setup used to obtain the shear stress vs shear rate data entered in the table. Fathom version 11 and earlier assumed all entered data was the Tube Flow Rheometer Data ($8V/D$) type. Refer to the Types of Rheological Data section below for more information on which option to use.

Note: For Herschel-Bulkley fluids only the Shear Rheometer Data option is available at this time. This is because the method employed in Fathom to convert from Tube Flow Rheometer Data to Shear Rheometer Data has unknown accuracy for use with the Herschel-Bulkley model.

Calculate Power Law Constants

Rheological Data

Raw Data Type: Tube Flow Rheometer Data (8V/D) Shear Rheometer Data (du/dy) ?

Parameter	Shear Rate 8V/D	Shear Stress Tau
Units	1/seconds	psia
1	20.86	0.0073
2	59.9	0.0082
3	78.82	0.0083
4	102.1	0.0086
5	127.7	0.0088
6	150.2	0.009
7	174.6	0.0092
8	201.8	0.0097
9	222.3	0.0106
10		
11		

Curve Fitting
Curve Fit Type: Polynomial

Shear Stress

Curve Fit Order: 1

Generate Curve Fit Now

Edit Table

OK Cancel Help

Figure 2: Rheological data input form

The user can click Generate Curve Fit Now to curve fit the data and obtain the viscosity model constants. Generating the curve fit will show the user a plot of their data along with the fitted constants, as shown below. Clicking OK will transfer those constants to the Viscosity Model panel and display the resultant Shear Stress vs Shear Rate curve fit graph.

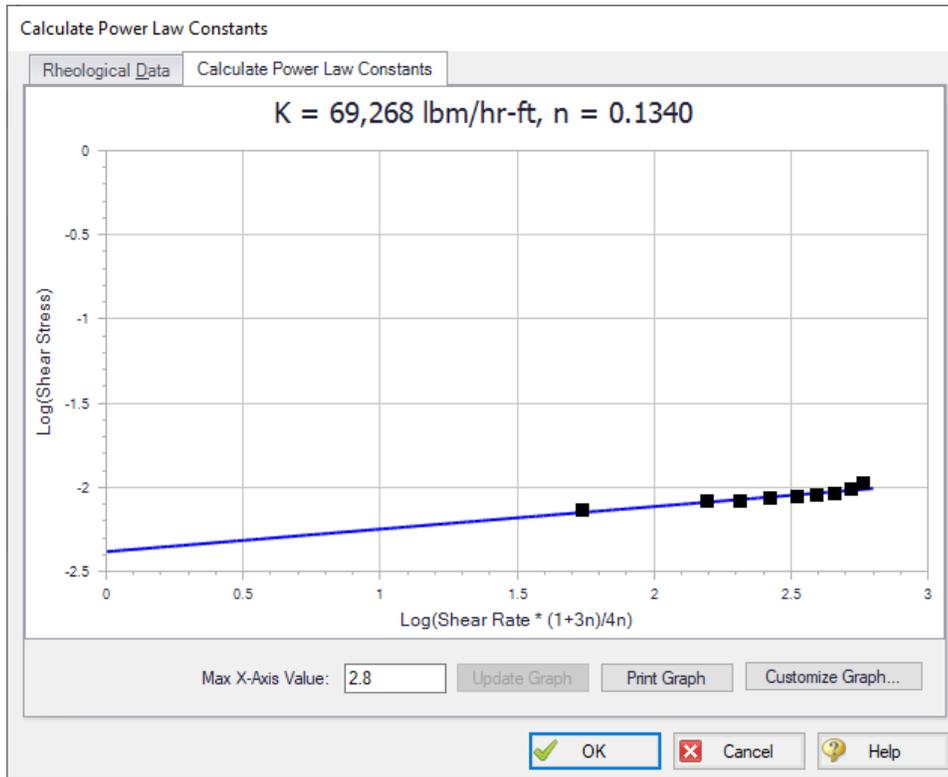


Figure 3: Curve fit rheological data and constants

Types of Rheological Data

Two major types of rheological test data exist. Many names and potential test devices exist for each type. For the purposes of this discussion, we must pick a single name for each type. One experimental setup is a tube flow rheometer while the other is a rotational rheometer.

Tube Flow Rheometer

A tube flow rheometer system consists of a pipe of some diameter and length connected to a high pressure source and a low pressure source. Varying the high pressure source drives flow through the pipe at different velocities. This experimental setup allows the engineer to obtain data on the pressure gradient (pressure drop per length) along the pipe for a given velocity. The engineer could also obtain this type of rheological data by directly measuring pressure and velocity along a pipe in an existing facility.

This data is then manipulated into shear stress and shear rate data, respectively. Specifically, the tube flow rheometer gives shear rate data where the shear rate is the bulk shear rate of the fluid. The measured velocity is the bulk velocity of the fluid, meaning only bulk shear rate can be calculated. Bulk shear rate can be calculated from bulk velocity using the equation $8V/D$.

This type of rheological data may be more common in industries considering slurry applications or when the process facility is already in operation and data is obtained via direct measurements of the system operations.

Rotational Rheometer

A rotational rheometer system consists of a rotating device in contact with the fluid being studied. Common examples of this system include the cup and bob (couette) rheometer and the cone and plate rheometer. Varying the angular rate of the rotating device creates different shear rates in the fluid. This experimental setup allows the engineer to obtain data on the torque required to achieve a given angular rate of rotation.

This data is then manipulated into shear stress and shear rate data, respectively. Specifically, the rotational rheometer gives shear rate data where the shear rate is the wall shear rate of the fluid. The angular rate of rotation and equipment geometry are directly known, and thus the shear rate is calculated at the wall of the rotating device.

This type of rheological data is more common when data is obtained via small-scale, laboratory tests. If rheological data is presented with information about rate of rotation (RPM, revolutions per second, Hz, etc.), the data was likely obtained from a rotational rheometer.

Finally, modern rotational rheometers will commonly manipulate the directly measured torque and angular rate of rotation data for the engineer, returning the raw data along with shear stress or viscosity vs. shear rate data. Note that Fathom requires the user to define shear stress vs. shear rate data when entering rheological test data. If the user only has viscosity vs. shear rate data, they should curve fit the constants for the desired viscosity model manually.

Converting Between Types of Rheological Data

Although two types of rheological data exist, most viscosity models expect rotational rheometer data. Fathom employs the Rabinowitsch-Mooney technique to convert from tube flow rheometer data to rotational rheometer data when necessary.

The Rabinowitsch-Mooney technique can be applied to obtain rotational rheometer data from tube flow rheometer data as follows:

1. Plot the tube flow rheometer data on a log-log scale.
2. Determine the slope of the data. The slope is represented by n' .
3. Multiply the tube flow rheometer data by the value:

$$\frac{3n' + 1}{4n'}$$

Fathom now has rheological data in the form of rotational rheometer (rheogram) data, which can be used to directly fit the viscosity model constants.

Related Topics

Viscosity Model Panel.....	97
Power Law Model.....	103
Bingham Plastic Model.....	106
Herschel-Bulkley Model.....	108

Related Examples

Non-Newtonian Phosphates Pumping	918
Non-Newtonian Mill Discharge Slurry	974

Heat Transfer/Variable Fluids Panel

The Heat Transfer/Variable Fluids panel, shown in Figure 1, allows users to define how the fluid properties will vary. The fluid selected on the Fluid panel must have data entered for the heat transfer parameters, such as specific heat, for the Heat Transfer options to be available, as is discussed further in the [Custom Fluids](#) topic.

Note: Heat transfer options may be disabled for several reasons:

1. Only [Newtonian fluids](#) can be modeled when heat transfer is enabled.
2. User Specified Fluids only contain data for density and viscosity, and thus heat transfer can't be modeled.
3. [User Library Fluids](#) can only model heat transfer if they contain data for specific heat, thermal conductivity, and a reference temperature/enthalpy point.

- **Constant Fluid Properties** - All fluid properties are the same for every pipe and are as specified on the Fluid panel. Note that [non-newtonian viscosity models](#) will still cause the viscosity to vary if the constant fluid properties option is selected.
- **Variable Fluid Properties** - The values for density and viscosity on the Fluid panel are the default fluid properties. You can enter [different fluid property values](#), if desired, for any pipe in the [Pipe Properties window](#). This does not mean that fluid properties can vary on their own in the model - they are constant in each pipe and must still be directly specified by the user.

This option may be useful to model different pipes carrying different fluids, or to model changes in temperature across a component. Fathom cannot perform dynamic mixing; however, fluid properties of pure components may be individually specified in upstream pipes, and the resulting mixture properties can be individually specified in downstream pipes.

- **Heat Transfer With Energy Balance (Single Fluid)** - All fluid properties are determined dynamically during the run based on the most recent [solution for temperature](#) and pressure - thus they are inherently variable. Only one fluid is used in the model. Additional inputs will be required in the pipes and junctions which are specified to [model heat transfer](#). If a User Library Fluid is utilized in the model, it will need to have data for specific heat, thermal conductivity, and a reference temperature/enthalpy point to allow heat transfer.
- **Heat Transfer With Energy Balance (Multiple Fluid)** - All fluid properties are determined dynamically during the run based on the most recent [solution for temperature](#) and pressure - thus they are inherently variable. Piping loops with different fluids can be modeled using [Fluid Groups](#) to input multiple fluids into the model, such as to represent different sides of a heat exchanger. However, dynamic mixing of fluids is not supported. Additional inputs will be required in the pipes and junctions which are specified to [model heat transfer](#).

Always Use Constant Density - Only applicable to *Variable Fluid Properties* and *Heat Transfer With Energy Balance (Single Fluid)*. This option forces the fluid density to remain constant throughout the model. This can be convenient when density variations due to temperature are minimal but viscosity changes are significant. Enabling this option has only minimal impact on results in these cases, and can

greatly simplify and clarify the head and gradeline values which would otherwise change by some minimal amount.

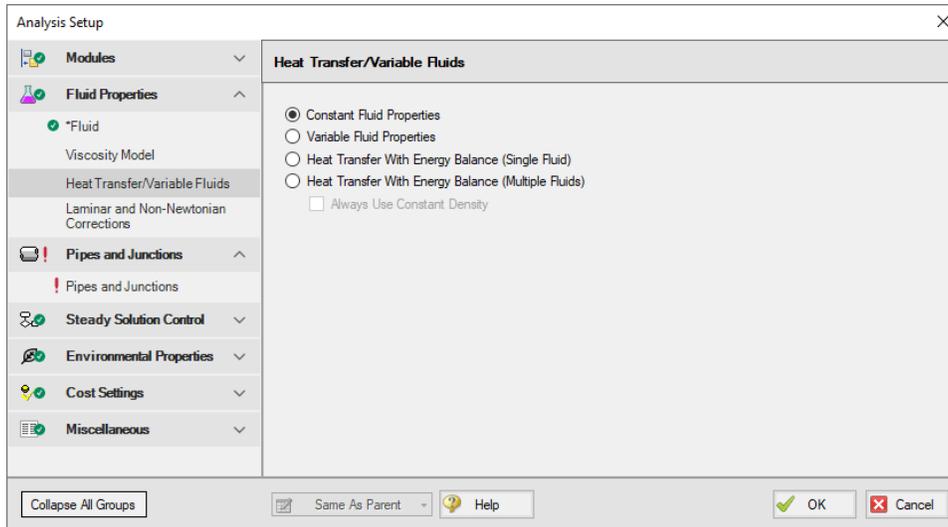


Figure 1: Heat Transfer/Variable Fluids panel

Related Topics

Fluid Properties Group.....	90
Fluid Panel.....	91
Viscosity Model Panel.....	97
Laminar and Non-Newtonian Corrections Panel.....	123
SSL Module Fluid Properties Group.....	131
Modeling Heat Transfer.....	121
Fluid Groups.....	122
HGL, EGL and head loss results may not be meaningful for variable density systems.....	779
Junction X has a density change across it.....	786

Related Examples

Freon Delivery System with Heat Transfer	888
--	-----

Modeling Heat Transfer

AFT Fathom can model heat transfer simultaneously with the hydraulic calculations. Since many pipe flow design calculations neglect heat transfer, this capability is turned off by default. Hence temperature and heat loss information is not required to specify a model.

Heat transfer modeling is enabled on the [Heat Transfer/Variable Fluids panel](#) by selecting one of the two Heat Transfer With Energy Balance options. Once selected, temperature input is required at boundary junctions such as [reservoirs](#). By default pipes are adiabatic, but different [heat transfer models](#) can be

selected such as [Convective Heat Transfer](#) and [Constant Heat Flux](#). With Convective Heat Transfer pipes can be assigned insulation and convection coefficient data to calculate heat loss or gain.

In addition, [heat exchangers](#) provide input for heat transfer with eleven [heat exchanger thermal models](#) available. Thermal linking with other heat exchangers is also provided to model cases such as heat transfer between both sides of a heat exchanger.

Finally, heat addition at pumps can also be modeled.

As heat transfer takes place during the system hydraulic solution, physical properties (such as viscosity) are dynamically updated based on resulting temperatures and a [system energy balance](#) is performed as part of the simulation.

Related Topics

Heat Transfer/Variable Fluids Panel.....	120
Fluid Groups.....	122

Related Examples

Freon Delivery System with Heat Transfer	888
--	-----

Fluid Groups

The purpose of fluid groups is to allow heat transfer calculations in different loops which have different fluids. When enabled, a new menu item called Fluids appears on the Edit menu Groups flyout menu. Here is how this feature is enabled and used:

1. On the [Fluid panel](#), select a base fluid
2. On the [Heat Transfer/Variable Fluids panel](#), select Heat Transfer With Energy Balance (Multiple Fluids)
3. Select each pipe in the fluid loop on the Workspace and go to the Edit menu, Groups option, then Create to give the group a name. At this point the pipes and junctions are standard AFT Fathom [groups](#).
4. Go to the Heat Transfer/Variable Fluids panel
5. Check the Use As Fluid Group box for the desired fluid group
6. Specify what fluid is to be used in each fluid group.
7. Click OK

Now when you run the model, each of the pipes and junctions in each group will take on the fluid properties assigned to that fluid group.

Note: If you are using a custom fluid in your model and make changes to it in the fluid library, you will need to save and re-open the model for the changes to be applied to any fluid in a Fluid Group.

Here are some guidelines for use:

- A pipe or junction can belong to one and only one fluid group
- Every pipe and junction must belong to a fluid group

Use with Heat Exchangers

Fluid groups allow [heat exchangers](#) to be modeled with thermal links to other heat exchangers carrying different fluids.

Thermal Links

When modeling convective heat exchangers (such as parallel or counterflow) two heat exchanger junctions can be linked thermally. A thermal link is when two [heat exchanger junctions](#), which are hydraulically isolated, can exchange heat with each other. Typical use is when the two heat exchangers represent two sides of the same physical heat exchanger. When thermal linking specified in a heat exchanger properties window, the user must specify the heat transfer area and overall convection coefficient. AFT Fathom will force the two heat exchanger junctions to use the same value.

Related Topics

Heat Transfer/Variable Fluids Panel.....	120
Modeling Heat Transfer.....	121

Laminar and Non-Newtonian Corrections Panel

AFT Fathom uses the methods and correlations discussed for each viscosity model only to model frictional losses for pipes. Frictional losses due to pipe fittings or junction losses are modeled using the methods discussed on this page regardless of viscosity model.

It is generally believed that the pressure drop through valves, fittings, etc derive from the physics of bulk fluid motion, wall separation, turbulence, etc and hence are not affected by Non-Newtonian fluid behavior in turbulent flow. Thus, pressure loss due to pipe fittings or junction-specific losses during turbulent flow are modeled using the same methods employed for Newtonian flow.

During laminar flow, pipe fittings and junction losses may not be similar for both Newtonian and Non-Newtonian fluids. AFT Fathom offers two options to model these losses during laminar flow. The correction options are:

- Use [Adjusted Turbulent K Factor Method \(ATKF\)](#) - derived from the Crane hydraulic handbook ([Crane 1988](#)).
 - Also Apply in Turbulent Region
 - Use [Improved Method](#)
- Use [Non-Newtonian to Newtonian Pipe Friction Ratio Multiplier](#)

Modifying Losses for Fittings and Junctions Interface

The options listed above to modify losses at fittings and junctions can be found on the Laminar and Non-Newtonian Corrections panel in the Fluids Properties group of the Analysis Setup, as shown in Figure 1 below.

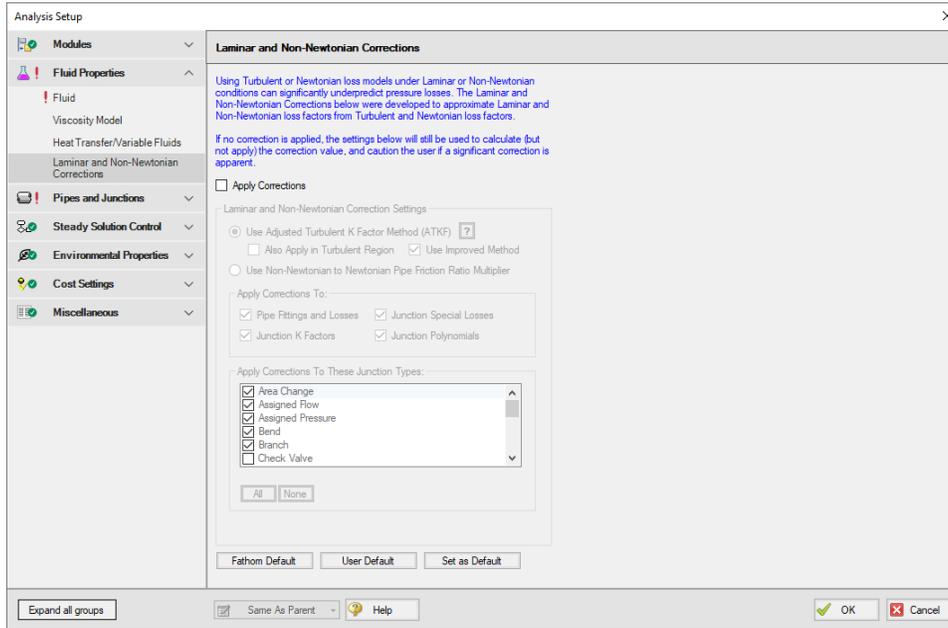


Figure 1: Options to modify fitting and junction losses on the Laminar and Non-Newtonian Corrections panel

These corrections are off by default and "Apply Corrections" must be selected to access them. Users can select which correction method is applied and can select which junction types the loss model will be applied to. Fathom Default can be selected at any time to reset the selected junction types.

Non-Newtonian to Newtonian Pipe Friction Ratio Multiplier

One approach to modeling losses associated with fittings and junction losses is to assume that the ratio of those losses in a Non-Newtonian and Newtonian fluid is equivalent to the ratio of pipe friction in a Non-Newtonian and Newtonian fluid. One can write:

$$K_{Non-Newtonian} = K_{Newtonian} \frac{f_{Non-Newtonian}}{f_{Newtonian}}$$

This approach uses the pipe friction at the adjacent upstream pipe and the uncorrected loss factor in a Newtonian fluid to model the losses associated with the fitting or junction for a Non-Newtonian fluid.

Related Topics

Fluid Properties Group.....	90
Fluid Panel.....	91
Viscosity Model Panel.....	97
Heat Transfer/Variable Fluids Panel.....	120
SSL Module Fluid Properties Group.....	131
Adjusted Turbulent K Factor Method (ATKF).....	125
Improved ATKF Method.....	127
Note about Laminar Corrections for Fittings.....	130
3-K (Darby) Method.....	266

Adjusted Turbulent K Factor Method (ATKF)

[Crane \(1988\)](#) includes an equivalence between equivalent length methods and K-factor on the nomograph chart on page A-30. This nomograph is based on theory as discussed below.

The basic law of pressure drop for friction factor is

$$dP = f \frac{L}{D} \left(\frac{\rho V^2}{2} \right) \quad (1)$$

For K-factors the pressure drop is given by

$$dP = K \left(\frac{\rho V^2}{2} \right) \quad (2)$$

From equations 1 and 2 it can be seen that there is an equivalence between f^*L/D and K .

The equivalent length method works by adding to the physical length as follows:

$$dP = f \frac{L + L_{eq}}{D} \left(\frac{\rho V^2}{2} \right) \quad (3)$$

Hence the pressure drop due to the methods can be related as follows:

$$K = f \frac{L_{eq}}{D} \quad (4)$$

An appropriate equivalent length can be determined from standard AFT Fathom K-factor data by looking at large Reynolds number and hence fully turbulent conditions

$$L_{eq,turb} = K_{turb} \frac{D}{f_{turb}} \quad (5)$$

And finally a general K-factor which is based on equivalent length can be found in the following:

$$K = K_{turb} \frac{f}{f_{turb}} \quad (6)$$

To apply the method, take the AFT Fathom K-factors (K_{turb}) and multiply them by the relevant pipe friction factor, f , at actual conditions (and Reynolds number) and divide them by the turbulent friction factor, f_{turb} , evaluated at very large Reynolds number.

Improved ATKF Method

For the [Improved ATKF method](#), the turbulent friction factor in equation 6 is replaced with the turbulent friction factor calculated for commercial steel at very large Reynolds numbers, which is about 0.023 for a roughness of 0.0018 in. This allows the adjusted K factor to be independent of pipe roughness in the laminar flow region so that it more closely mimics the expected behavior for the 3K method from [Darby](#) and the [equivalent length](#) method.

Comparison of Equation 6 to Published Data

A study was performed to evaluate how Equation 6 compares to published data ([Wood 2012](#)). A comparison for a 10 foot long, 2 inch in diameter pipe flowing water was performed with the following:

1. No fittings in the pipe
2. Fittings of 12 elbows of 1.5 r/D with a total K of 3.17 but no corrections (this is the way standard AFT Fathom works)
3. Darby 3-K method for 12 elbows (see [Darby, 2001](#), pp.209)
4. Equivalent length of 12 elbows with $L_{eq}/D = 16$ (see Darby, 2001, pp.209)
5. Use Equivalent Length in Pipe Diameters (L/D) Derived From K-Factors, Eq. 6

Figure 1 shows the results on a log-log plot. One would not expect the first case to match the others as no fittings were included. Hence the head gradient was less than the other cases. However, Case 1 does illustrate how Case 2 works in the standard AFT Fathom. Case 2 agrees with Cases 3-5 at high Reynolds Number but agrees with Case 1 at low Reynolds number. This is a result of using turbulent, high Reynolds Number K-factors for low Reynolds Number applications.

Cases 3, 4 and 5 all agree well throughout the entire range of Reynolds Numbers. There is some small discrepancy at high Reynolds Number which is merely a result of various reference differences over the exact pressure loss through a 1.5 r/D elbow.

The conclusion from Figure 1 is that the Equation 6 method allows reliable adjustment of turbulent pressure loss data for fittings and junctions to laminar applications.

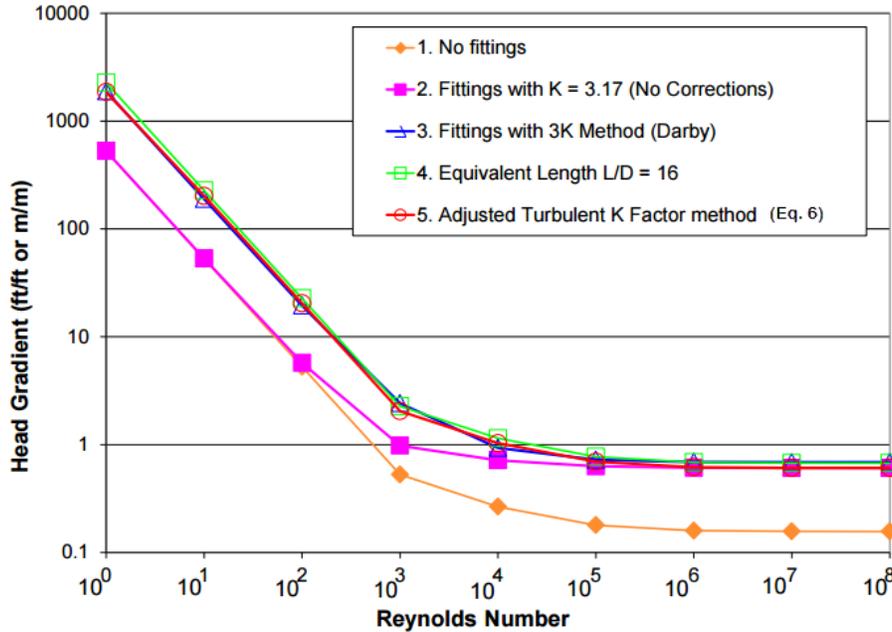


Figure 1: Log-log plot of dimensionless head gradient vs. Reynolds number for various fitting loss models

Related Topics

Laminar and Non-Newtonian Corrections Panel	123
Improved ATKF Method	127
Note about Laminar Corrections for Fittings	130
3-K (Darby) Method	266

Improved ATKF Method

For commercial steel pipe and pipes with a similar roughness, it has been shown that the equivalent length, Darby 3-K, and Adjusted Turbulent K Factor (ATKF) methods will all produce similar results in the laminar flow region (see [Adjusted Turbulent K Factor \(ATKF\) Method](#)).

However, this is not the case when the pipe roughness varies significantly from that of commercial steel. At different roughness values the predicted head loss with the original ATKF method varies inversely with pipe roughness, so that smoother pipes will experience higher head loss in the laminar regime as is shown in Figure 1. The Darby 3-K method shows no dependence on pipe roughness in the laminar regime, as is shown in Figure 2. It would be expected that head loss in the laminar regime would behave as predicted by the 3-K method, based on the fact that the laminar friction factor is independent of pipe roughness.

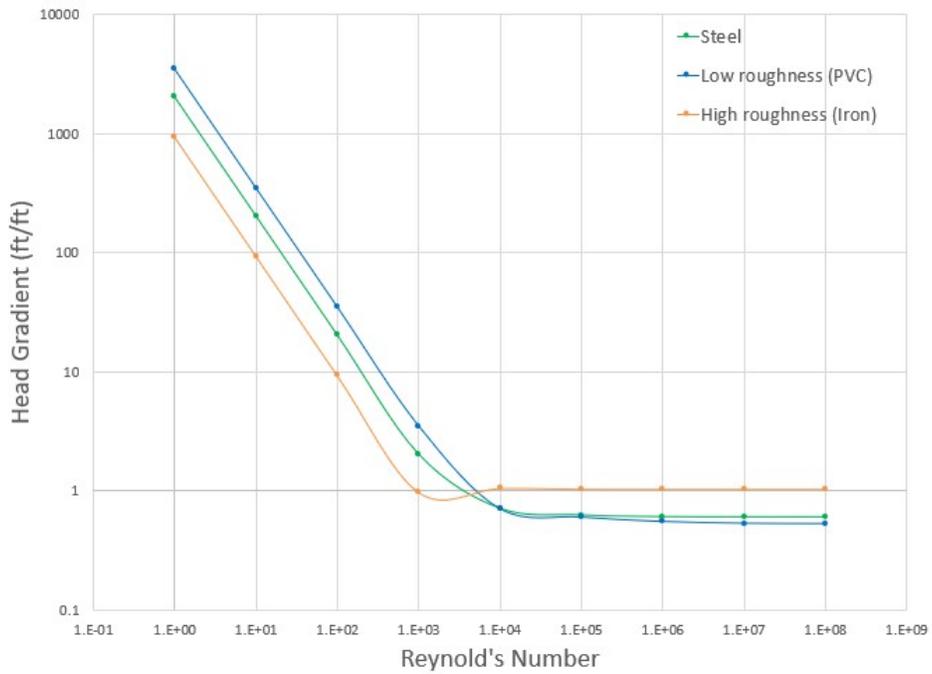


Figure 1: Head Gradient calculated with the original ATKF method at different roughness values, showing differences in the laminar region

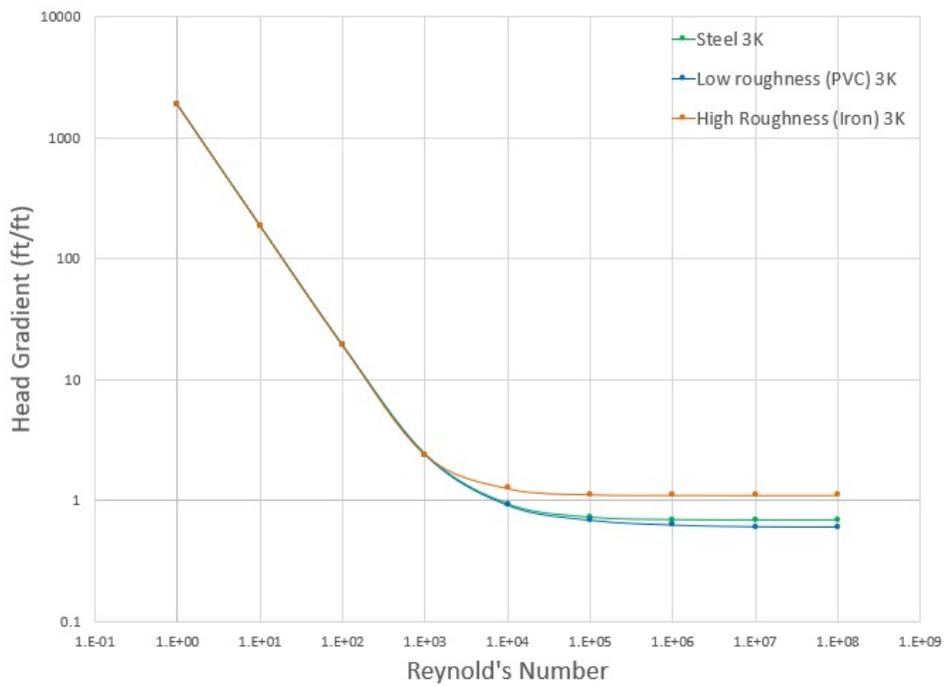


Figure 2: Head Gradient calculated with the 3-K method at different roughness values, showing no differences in the laminar region

To remove the dependence on pipe roughness in the ATKF method, the Improved ATKF method uses a fixed turbulent friction factor value calculated for commercial steel pipe (roughness 0.0018 in) at very large Reynolds numbers. The Improved ATKF method can be written as follows, replacing f_{turb} with the reference turbulent friction factor, $f_{turb,ref}$:

$$K = K_{turb} \frac{f}{f_{turb,ref}}$$

With the Improved ATKF factor it can be seen that the head loss is no longer dependent on pipe roughness (Figure 3).

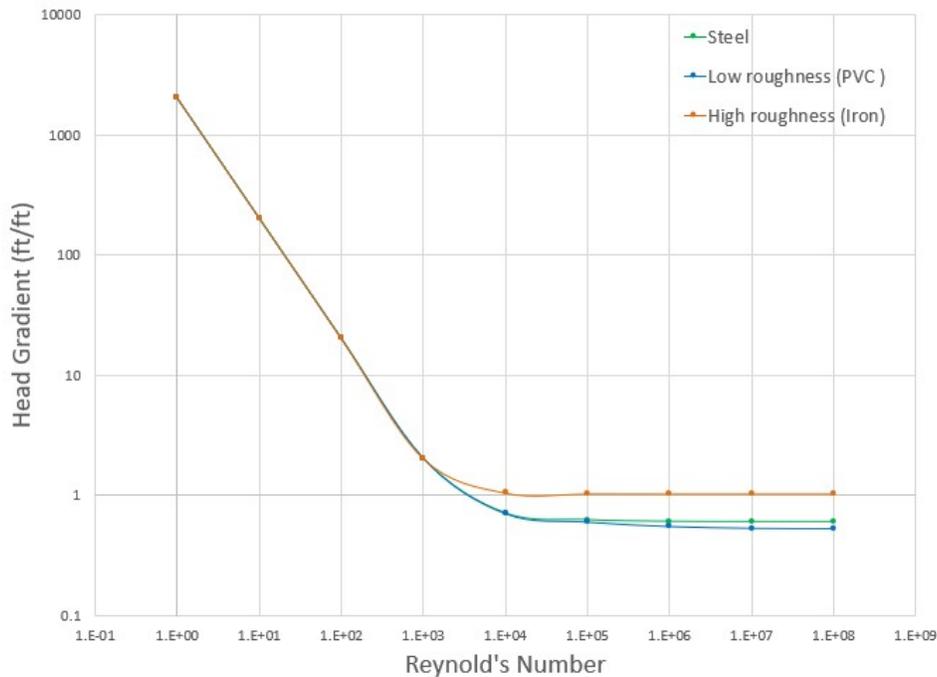


Figure 3: Head gradient calculated with the Improved ATKF method at different roughness values

Related Topics

Laminar and Non-Newtonian Corrections Panel	123
Adjusted Turbulent K Factor Method (ATKF)	125
Note about Laminar Corrections for Fittings	130
3-K (Darby) Method	266

Note about Laminar Corrections for Fittings

There is ambiguity in the literature about how to calculate pressure loss for fittings with laminar flow. Here is a quick summary of different views:

No Laminar Correction Required

[Crane \(1988\)](#) suggests no correction is required. On pages 2-8 through 2-11 Crane discusses this. Worked examples 4-7, 4-8 and 4-9 use this assumption.

This assumption is the same as method 2 in the [ATKF comparison section](#) and Figure 1 (in the aforementioned topic), and is consistent with how AFT Fathom has always performed laminar calculations. This assumption is available by merely not applying the laminar methods discussed in this chapter.

Laminar Correction is Required

Publications such as [Darby \(2001\)](#) discuss the necessity for laminar corrections for fitting K-factors. On pages 209-214 “2-K” and “3-K” methods are discussed that purport to cover the entire range of Reynolds number.

Based on [Equation 6](#) and Crane’s (1988) nomograph of page A-30 a laminar correction based on equivalent length methods can be applied. This gives similar pressure drop results as the 3-K method and is available through the [Adjusted Turbulent K Factor method](#).

Low Reynolds Number Flow

In addition to the discussion above, there is a separate debate over whether flow at very low Reynolds numbers (less than 100) should be treated separately when determining if corrections are needed for fittings and losses. There are conflicting CFD studies and lab tests showing that corrections are needed, or are not needed. For additional details please see the below journal articles:

- **No corrections needed at low Re**
 - Liou, R. J., et.al, “Numerical Analysis of Steady Flow in a 90-deg Circular Section Bend”, J. Eng. Mech., v. 114 (1988), p. 1871.
 - Berger, S. A., et.al, “Flow in Curved Pipes”, Ann. Rev. Fluid Mech., v. 15 (1983), p. 461.
- **Corrections needed at low Re**
 - Edwards, M. F., et.al, “Head Losses in Pipe Fittings at Low Reynolds Numbers”, Chem. Eng. Res. Des., v. 63, (Jan 1985), p. 43.
 - Kittredge, C. P. and Rowley, D. S., “Resistance Coefficients for Laminar and Turbulent Flow Through One-Half-Inch Valves and Fittings”, Trans. ASME, v. 79, (Nov. 1957), p. 1759.

Related Topics

Laminar and Non-Newtonian Corrections Panel.....	123
Adjusted Turbulent K Factor Method (ATKF).....	125
Improved ATKF Method.....	127
3-K (Darby) Method.....	266

SSL Module Fluid Properties Group

([SSL Module Only](#)) When the Settling Slurry (SSL) Module is activated on the [Modules panel](#), the items in the Fluid Properties Group change to allow slurry physical property calculation.

Engineers who are new to slurry calculations may not recognize some of the properties in the following panels. These properties all relate to the theory of slurry calculations, which is discussed in the [Slurry Calculation Methods](#) section.

- [Carrier Fluid panel](#) - Define the Carrier Fluid Properties
- [Solids Definition panel](#) - Define the Slurry Calculation Method and Solids Specifications such as M (Stratification Ratio Exponent), Terminal Velocity Parameter, density, d_{50} , and d_{85}
- [Slurry Definition panel](#) - Define the Slurry Model and Slurry Properties

Related Topics

Fluid Properties Group.....	90
Fluid Panel.....	91
Viscosity Model Panel.....	97
Heat Transfer/Variable Fluids Panel.....	120
Laminar and Non-Newtonian Corrections Panel.....	123
SSL - Settling Slurry.....	550
Carrier Fluid Panel.....	131
Solids Definition Panel.....	133
Slurry Definition Panel.....	136
SSL Nomenclature.....	570

Related Examples

Beginner - Pump Sizing for Sand Transfer System - SSL.....	1092
Slurries with Variable Fluid Properties - SSL.....	1102
Slurry System Feasibility Study - SSL (Metric Only).....	1114

Carrier Fluid Panel

([SSL Module Only](#)) On the Carrier Fluid panel, users select and define the carrier fluid for the slurry. The available options are:

- Basic (Water)
- Advanced

Basic (Water) Carrier Fluid

The Basic (Water) option uses the Water (liquid) fluid from the [AFT Standard fluid library](#) as the carrier fluid and treats all pipes as having constant slurry properties (Figure 1).

Specify the water temperature for the slurry property calculation.

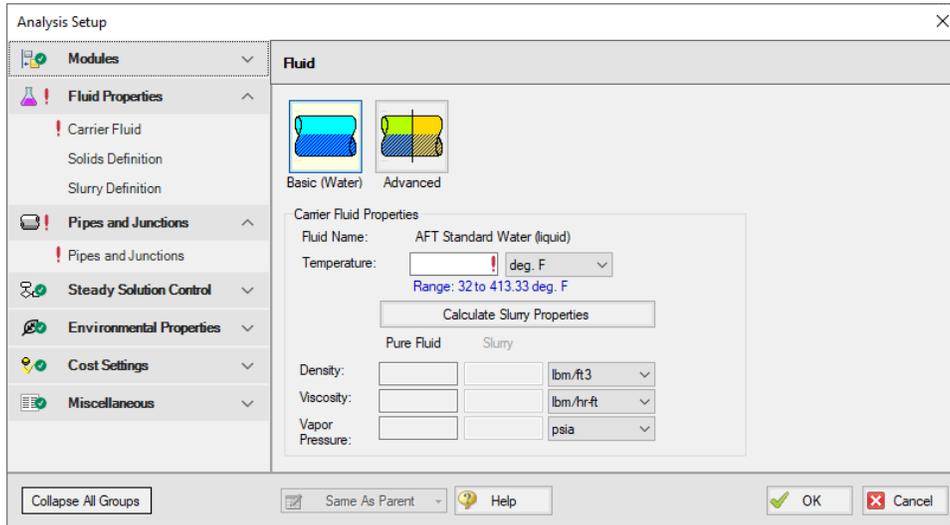


Figure 1: Carrier Fluid panel with the Basic (Water) carrier fluid option selected

Advanced Carrier Fluid

The Advanced option allows the user to specify any fluid as the carrier fluid, and will allow the slurry properties to vary among pipes.

The Carrier Fluid panel looks very similar to the [Fluid panel](#) when the SSL module is not activated. Specify the fluid, its temperature (and pressure in some cases), and fluid property variation on this panel. Users can access data from all fluid libraries. The User Specified Fluid option is disabled because it is not saved in a library. Custom fluids should be saved to a library and selected using the User Library Fluid option.

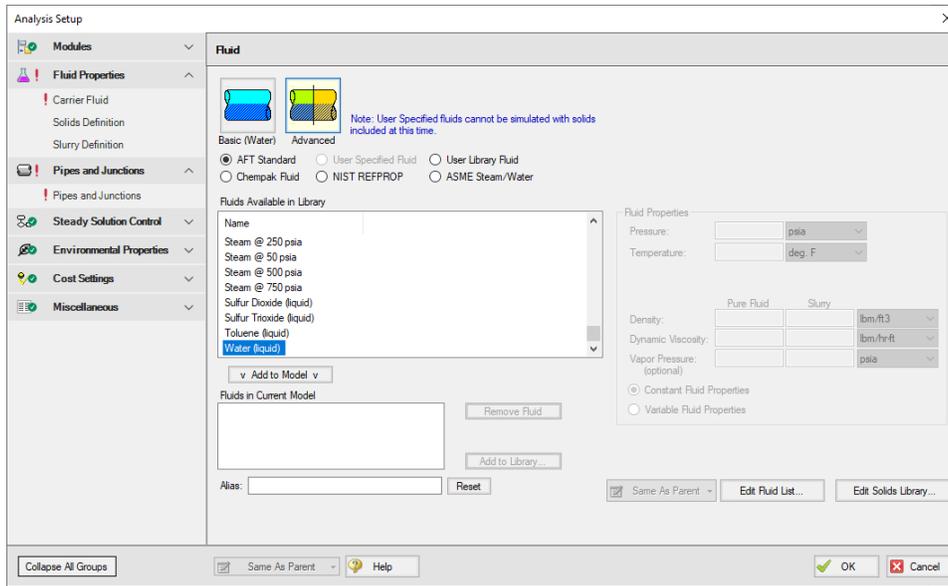


Figure 2: Carrier Fluid panel with the Advanced option selected

Related Topics

SSL - Settling Slurry.....	550
SSL Module Fluid Properties Group.....	131
Solids Definition Panel.....	133
Slurry Definition Panel.....	136
SSL Nomenclature.....	570

Related Examples

Beginner - Pump Sizing for Sand Transfer System - SSL.....	1092
Slurries with Variable Fluid Properties - SSL.....	1102
Slurry System Feasibility Study - SSL (Metric Only).....	1114

Solids Definition Panel

([SSL Module Only](#)) On the Solids Definition panel, users define the Slurry Calculation Method and Solids Specifications. These inputs are the same for both the Basic (Water) and Advanced [carrier fluid options](#).

Override options are provided for a number of parameters that allows users to modify data pulled from libraries should modeling requirements need this.

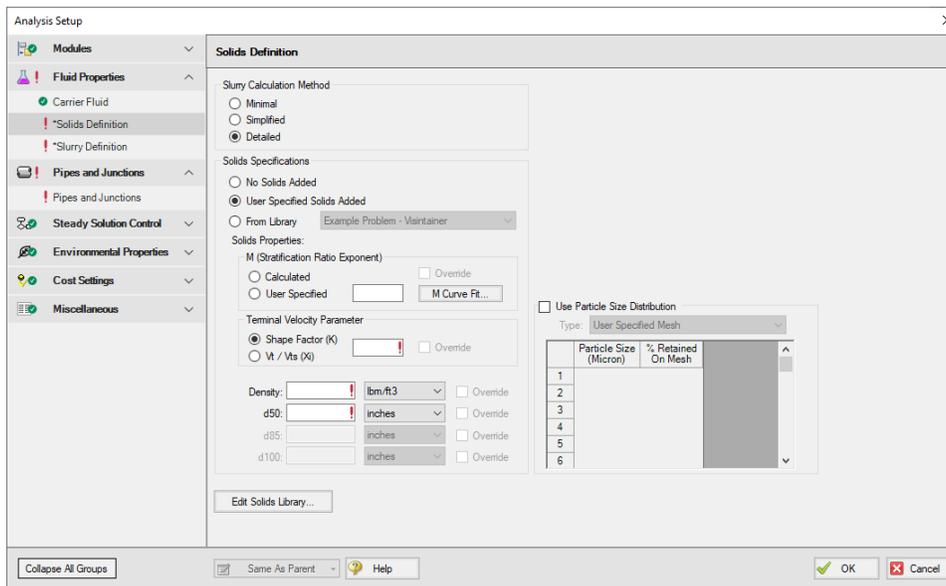


Figure 1: Solids Definition panel when Detailed Slurry Calculation Method and Calculated M are selected

Slurry Calculation Method

There are three calculation methods available to model slurries when using the Wilson, Addie, Clift slurry model. These are all variations of the Wilson, Addie, Clift slurry approach, and this selection is only applicable when Settling Slurry - Wilson, Addie, Clift is chosen as the active Slurry Model in the Slurry Definition panel. Each method requires different amounts of input data resulting in the use of different calculations. These calculations are described in the [Slurry Calculation Methods](#) section.

- [Minimal Method](#) - Used in the preliminary design phase where only rough data are available. The solids density, sliding friction coefficient, and solids concentration are the only parameters needed. However, due to the minimal data the calculations should only be used for early stage approximations because it often over-predicts losses. This is a conservative method that assumes a fully-stratified sliding bed.
- [Simplified Method](#) - Additional information is required and gives a better solution. This method requires the solids density, d50 particle size, a user-specified M exponent, and the solids concentration. This method assumes heterogeneous flow with suspended particles. This is a more accurate model than the Minimal method because it accounts for more slurry definition.
- [Detailed Method](#) - The default and provides the most rigorous solution methodology. The solids density, d50 particle size, terminal velocity parameter, and the solids concentration are required. Additionally, if the M exponent is to be calculated, the d85 particle size is also required. As with the Simplified Method, this method also assumes heterogeneous flow with suspended particles.

Solids Specifications

In the Solids Specification area, select from the following options for solids:

- No Solids Added - The model will run using the carrier fluid without any solids, no further input is required.

Note: The No Solids Added option is only compatible with the Newtonian Slurry Model option on the [Slurry Definition panel](#). The other Slurry Models imply that solids are present in the fluid, therefore, the Slurry Definition panel will be undefined until the Solids Specifications is changed to one of the options below.

- User Specified Solids Added - Define the solids using the Solid Properties fields.
- From Library - Use solids already defined in the Solids Library. Note no default solids library is provided by AFT, library items must be defined first in the [Library Manager](#) to be available here.

Solids Properties

By selecting User Specified Solids Added, the Solids Properties fields become available to define the solids. Depending on the Slurry Calculation Method selected at the top of the Solids Definition panel, certain fields are required:

- Minimal
 - Density
- Simplified
 - User Specified M (Stratification Ratio Exponent)
 - Density
 - d_{50}
- Detailed
 - M (Stratification Ratio Exponent)
 - Terminal Velocity Parameter
 - Density
 - d_{50}
 - d_{85} - if Calculated M (Stratification Ratio Exponent) is selected

The **M (Stratification Ratio Exponent)** is a parameter in the stratification ratio equation. The user can allow AFT Fathom to calculate M based on best available methods, or input M directly. The process of [estimating \$M\$ from recorded data](#) is discussed elsewhere.

The **Terminal Velocity Parameter** must be entered when using the Detailed calculation method. This can be entered as either a Shape Factor (K) or the ratio of the particle's terminal velocity (V_t) to that of a sphere (V_{ts}), designated as V_t / V_{ts} (X_i). If a Shape Factor is entered, it is used in conjunction with the graph shown in [Wilson, et al. 2006](#), on page 47, to obtain a value for X_i . Typical shape factors are given in Wilson, et al, 2006, on page 46 where $K = 0.26$ for sand.

In the lowest area, enter the required values for mean density of the solid particles, d_{50} (mass-median particle diameter) and d_{85} (diameter for which 85% by mass of the particles are finer).

4-Component Slurry Model

The [4-Component Slurry Model](#) requires a d100 and Particle Size Distribution.

Related Topics

SSL - Settling Slurry.....	550
SSL Module Fluid Properties Group.....	131
Carrier Fluid Panel.....	131
Slurry Definition Panel.....	136
SSL Nomenclature.....	570
Velocity in pipe X is less than the deposition velocity.....	784

Related Examples

Beginner - Pump Sizing for Sand Transfer System - SSL.....	1092
Slurries with Variable Fluid Properties - SSL.....	1102
Slurry System Feasibility Study - SSL (Metric Only).....	1114

Slurry Definition Panel

[\(SSL Module Only\)](#) The Slurry Definition panel allows selection of the Slurry Model and related options such as the Slurry Calculation Method, as well as input of solids concentration relative to the carrier fluid and estimation of bulk slurry properties, as shown in Figure 1.

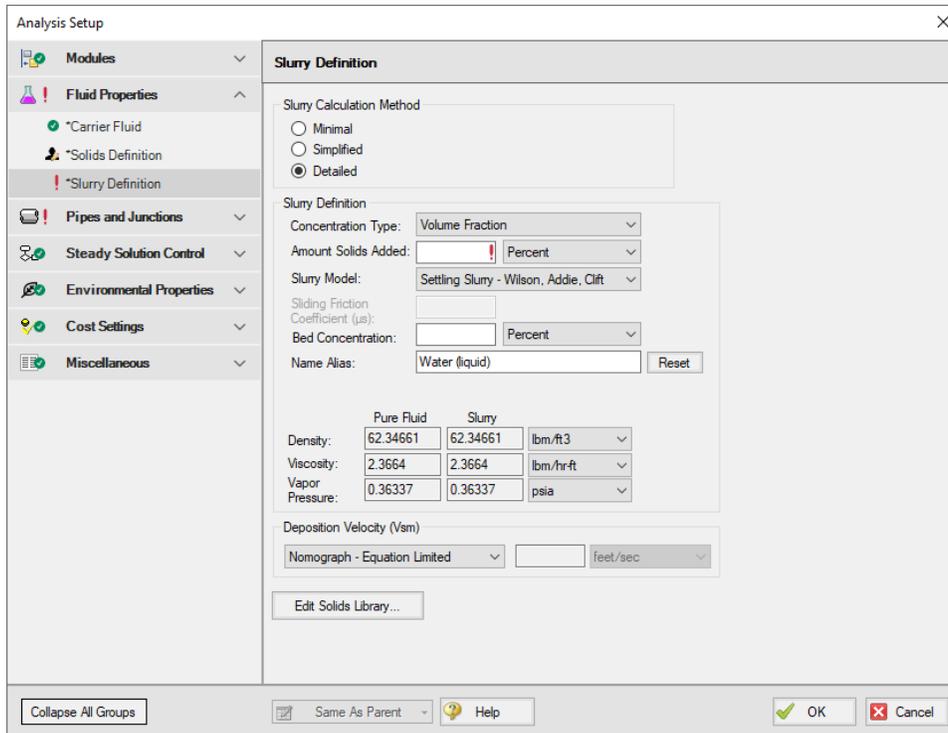


Figure 1: Slurry Definition panel when Detailed Slurry Calculation Method and User Specified Solids Added are selected

Slurry Calculation Method

There are three calculation methods available to model slurries when using the Wilson, Addie, Clift slurry model. These are all variations of the Wilson, Addie, Clift slurry approach, and this selection is only applicable when Settling Slurry - Wilson, Addie, Clift is chosen as the active Slurry Model in the Slurry Definition panel. Each method requires different amounts of input data resulting in the use of different calculations. These calculations are described in the [Slurry Calculation Methods](#) section.

- [Minimal Method](#) - Used in the preliminary design phase where only rough data are available. The solids density, sliding friction coefficient, and solids concentration are the only parameters needed. However, due to the minimal data the calculations should only be used for early stage approximations because it often over-predicts losses. This is a conservative method that assumes a fully-stratified sliding bed.
- [Simplified Method](#) - Additional information is required and gives a better solution. This method requires the solids density, d50 particle size, a user-specified M exponent, and the solids concentration. This method assumes heterogeneous flow with suspended particles. This is a more accurate model than the Minimal method because it accounts for more slurry definition.
- [Detailed Method](#) - The default and provides the most rigorous solution methodology. The solids density, d50 particle size, terminal velocity parameter, and the solids concentration are required.

Additionally, if the M exponent is to be calculated, the d85 particle size is also required. As with the Simplified Method, this method also assumes heterogeneous flow with suspended particles.

Slurry Definition

The **Concentration Type** is used to specify a value in the **Amount Solids Added** field. The available options for Concentration Type are:

- Grams per Liter of Mixture
- Volume Fraction
- Mass Fraction
- Solids Ratio (M-solids/M-liquid)

There are a few **Slurry Model** options to choose from. These options determine the calculations used in the Solver.

- **Newtonian** - Functionally disables most of the SSL module calculations and preserves only the viscosity properties of the fluid in the model.

Note: The No Solids Added option from the [Solids Definition panel](#) is only compatible with the Newtonian Slurry Model option. If you have already specified solids on the Solids Definition panel, you must select one of the other Slurry Models below.

- **Settling Slurry - Wilson, Addie, Clift** - Default option, applies slurry calculations from the Wilson, Addie, Clift Slurry Models.
- **Settling Slurry - 4-Component Model** - The [4-Component Slurry Model](#) developed by GIW Industries Inc. - A more complex model that takes into consideration the contribution of different size particles in the slurry flow.
- **Wasp-Durand Heterogeneous Model** - The [Wasp-Durand Slurry Model](#) is a two-layer slurry approach with heterogeneous and homogenous fluid components.

The **Sliding Friction Coefficient (μ_s)** is the coefficient of mechanical sliding friction between the granular mass and the boundary - where the boundary is where the motion takes place between the mass and a rigid boundary, i.e. the pipe wall. As the Sliding Friction Coefficient increases, so does the friction and thereby the pressure drop through the pipe. Typical values range from 0.4 to 0.5. This field is only available when the Wilson, Addie, Clift [Minimal Slurry Calculation Method](#) or the 4-Component slurry model are selected.

The **Bed Concentration (C_{vb})** is an optional parameter for the Wilson, Addie, Clift slurry methods and is only used to determine the actual settling velocity (V_s) shown in the output. These calculations are described in [Wilson \(1986\)](#) and are functions of the pipe diameter, particle size (d_{50}), particle density and relative concentration. If no value is entered for this parameter then a default of 0.60 is used. The maximum value of these settling velocities (V_s) is denoted V_{sm} and is the usual parameter of interest in designing a system to avoid operating with a stationary bed. This field is only available when the Simplified and Detailed Slurry Calculation Methods are specified at the top of the panel.

Deposition Velocity (V_{sm})

The Deposition Velocity (V_{sm}) can be determined using one of the options below. This is useful when the calculation method yields a slightly different value than, say, test results.

- **Nomograph - Equation Limited** (Wilson, Addie, Clift only) - This option is the default and uses the nomograph from [Wilson \(1986\)](#) page 111. The nomograph is noted to be conservatively high by Wilson, so an equation that calculates the max V_{sm} is used to limit the value of V_{sm} .

The results calculated from the nomograph will be checked against Equation 5.11 from Wilson (1986) as is shown below. If the value from the nomograph exceeds $V_{sm,max}$ calculated from Equation 5.11, then the $V_{sm,max}$ from Equation 5.11 will be used instead.

$$\frac{V_{sm,max}}{\sqrt{2gD(S_m - S_f)}} = \left(\frac{0.018}{f} \right)^{0.13}$$

Note: If the V_{sm} is calculated as a negative value using Equation 5.11 from Wilson then the V_{sm} will be displayed as 0 in the Output. If the V_{sm} is shown as 0, the inputs for the fluid and solid should be verified, and the user should consider calculating the V_{sm} using a separate method.

- **Equation Only** (Wilson, Addie, Clift only) - This option will cause the V_{sm} to be calculated solely using Equation 5.11.
- **User Specified** (Wilson, Addie, Clift only) - This option allows the user to directly specify V_{sm} .

Note: As Wilson describes there are times where operating at a velocity lower than V_{sm} is economically attractive and is without a stationary bed. These cases include a system where the solid concentration at the desired operating point is very different than that at the V_{sm} conditions. Wilson points out that this is rarely the case, and so the use of V_s instead of V_{sm} should be done with caution.

- **Nomograph Only** (4-Component only) - This option will cause deposition velocities to be calculated from the Wilson, Addie, Clift nomograph, similar to the Nomograph - Equation Limited Wilson, Addie, Clift option without the limiting equation. The other notable difference is that two deposition velocities are calculated for the 4-Component method from two different diameters; $V_{sm,h}$ from d_{50h} and $V_{sm,s}$ from d_s .
- **Match Heterogeneous Calculation** (Wasp-Durand only) - The calculation method for Wasp-Durand and deposition velocities is coupled to the selection of the Heterogeneous Gradient Calculation Method. The following methods are used, which are described in more detail in [Wasp-Durand Heterogeneous Slurry Model](#):
 - Wasp-Durand
 - Liu Dezhong
 - Fei Xiangjun

Wasp-Durand Options

When using the Wasp-Durand Slurry model, the Wasp-Durand Options section will be displayed within the Slurry Definition panel. Equations associated with these options may be found in the [Wasp-Durand Heterogeneous Slurry Model](#) section.

The **Sedimentation Velocity Model** determines which model is used to calculate sedimentation velocity, which is calculated per particle size group and is a particle settling velocity. This is the terminal particle freefall velocity that occurs when the drag, gravity, and buoyancy forces on a particle are balanced. There are two options available. Either option is appropriate for a wide range of flow states and particle sizes, including laminar, transitional, and turbulent flow, so the selection of model will depend on personal preference or industry standards:

- Liu Dezhong
- Ruby & Zanke (1977)

The **Heterogeneous Gradient Calculation Method** determines the set of equations to use for calculating the heterogeneous component of the slurry pressure gradient, along with the deposition velocity, as these options are always coupled for Wasp-Durand. Three options are available:

- Wasp-Durand
- Liu Dezhong
- Fei Xiangjun

The **Apply Apparent Slurry Properties** option allows the viscosity of the homogeneous (vehicle) component to be affected by solids, or specified by the user manually. Without this option enabled, the clear fluid carrier fluid viscosity will be used. The following viscosity options are available when applying apparent slurry properties:

- **User Specified Viscosity** - The viscosity used for the homogeneous pressure gradient is specified by the user rather than being inherited from the carrier fluid or calculated from the carrier fluid and particle size distribution. This option treats the homogeneous component as Newtonian.
- **Apply Viscosity Model** - These options calculate the carrier fluid viscosity including the contribution of solids. Solids of all sizes will contribute to the homogenous viscosity, but finer particles contribute more significantly.
 - **Thomas (1965)** - This can be used to calculate a Newtonian viscosity value or Bingham stiffness factor for each pipe.
 - **Fei Xiangjun - Mineral** - This method calculates a Newtonian viscosity or Bingham stiffness factor for each pipe, and can also calculate a global yield stress value for the slurry.
 - **Fei Xiangjun - Coal** - This method is mathematically identical to the Fei Xiangjun - Mineral viscosity models, but the empirical values are tuned to coal slurry data rather than mineral slurries.

When active, the **Use Hanks' Bingham Plastic Friction Model** checkbox replaces the Newtonian homogeneous component friction calculations with a non-Newtonian Bingham plastic approximation. With this option, yield stress may be manually specified or calculated using the Fei Xiangjun methods. The Bingham stiffness factor is calculated iteratively for each pipe using the Fei Xiangjun or Thomas (1965) methods.

Related Topics

SSL - Settling Slurry.....	550
SSL Module Fluid Properties Group.....	131
Carrier Fluid Panel.....	131
Solids Definition Panel.....	133
SSL Nomenclature.....	570

Related Examples

Beginner - Pump Sizing for Sand Transfer System - SSL.....	1092
Slurries with Variable Fluid Properties - SSL.....	1102
Slurry System Feasibility Study - SSL (Metric Only).....	1114

Pipes and Junctions Group

The Pipes and Junctions Group in [Analysis Setup](#) has one item: the [Pipes and Junctions panel](#). When the Pipes and Junctions group has a green checkmark next to its name, all pipes and junctions are fully defined. When a red exclamation point is next to its name, there is at least one pipe or junction that is undefined. The model cannot run until all pipes and junctions are defined.

Related Topics

Analysis Setup Overview.....	83
Modules Group.....	84
Fluid Properties Group.....	90
Transient Control Group.....	143
Goal Seek and Control Group.....	148
Steady Solution Control Group.....	162
Environmental Properties Group.....	168
Cost Settings Group.....	170
Automatic Sizing Group.....	173
Miscellaneous Group.....	175
Pipes and Junctions Panel.....	141

Pipes and Junctions Panel

The Pipes and Junctions panel within Analysis Setup displays the status of pipes and junctions within the model.

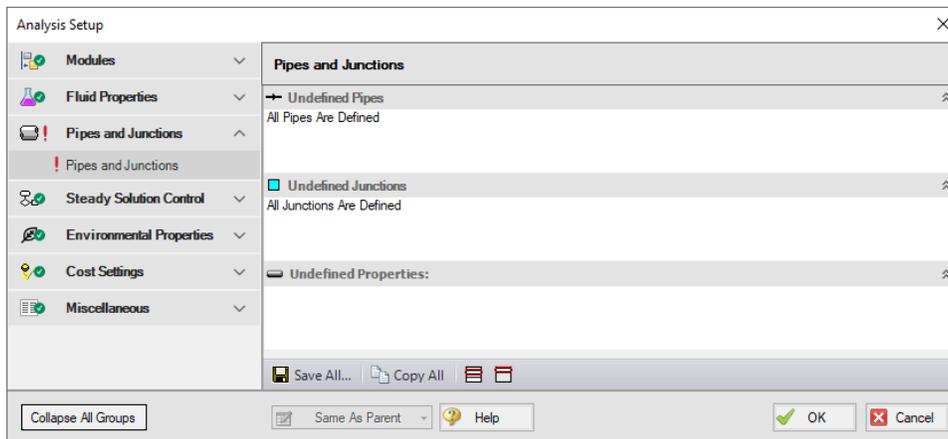


Figure 1: Default state of the Pipes and Junctions panel

This panel is made up of three sections:

- Undefined Pipes
- Undefined Junctions
- Undefined Properties

The Undefined Pipes and Undefined Junctions sections displays the number and name of all pipes and junctions, respectively, that are not fully defined. If all pipes and junctions in the model are fully defined, the messages "All Pipes Are Defined" and "All Junctions Are Defined" will be displayed. Clicking on the undefined pipe or junction will display the undefined properties in the list below, as shown in Figure 2. All undefined data for all objects can be saved to the file or copied to the clipboard. Any undefined object's Properties window can be launched by double-clicking the object's name in the list to finish defining the object.

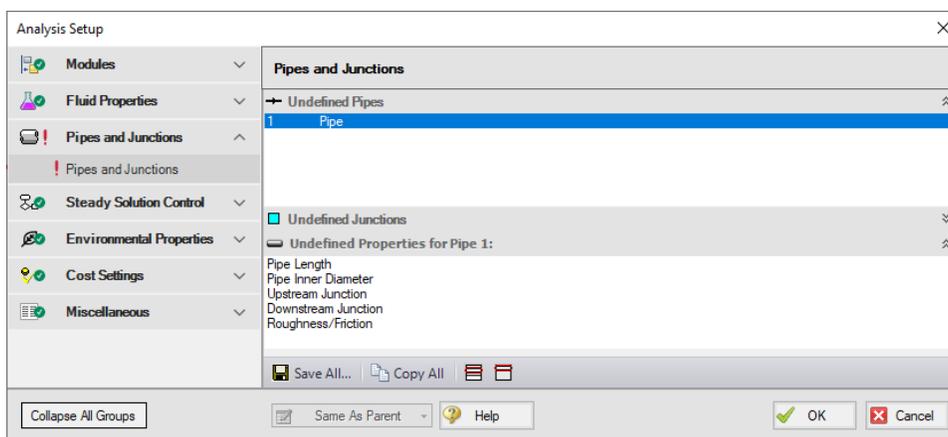


Figure 2: Pipes and Junctions panel with one pipe undefined and undefined properties displayed

Related Topics

Pipes and Junctions Group.....	141
--------------------------------	-----

Transient Control Group

The Transient Control Group in [Analysis Setup](#) is where settings pertaining to a transient simulation using the [XTS Module](#) are managed. The Transient Control Group will have a green checkmark next to its name when the XTS Module is disabled (Steady Only mode) or when all required fields on the [Simulation Duration panel](#) are defined. The model cannot run until the Transient Control Group has a green checkmark.

Related Topics

Analysis Setup Overview.....	83
Modules Group.....	84
Fluid Properties Group.....	90
Pipes and Junctions Group.....	141
Goal Seek and Control Group.....	148
Steady Solution Control Group.....	162
Environmental Properties Group.....	168
Cost Settings Group.....	170
Automatic Sizing Group.....	173
Miscellaneous Group.....	175
XTS - Extended Time Simulation.....	585
Simulation Duration Panel.....	143

Related Examples

Beginner - Filling a Tank - XTS	1038
Reservoir Balance - XTS	1050
Valve Closing - XTS	1058
Housing Project - XTS	1072
Variable Demand - XTS	1081
Controlled Heat Exchanger Temperature - XTS	1066
Fixed Head Supply Tank - GSC XTS	1263

Simulation Duration Panel

[\(XTS Module Only\)](#) When the XTS module is activated, the Transient Control group appears in Analysis Setup. Whether the XTS module is enabled (Transient mode) or disabled (Steady Only mode), the Simulation Duration panel can be opened and modified.

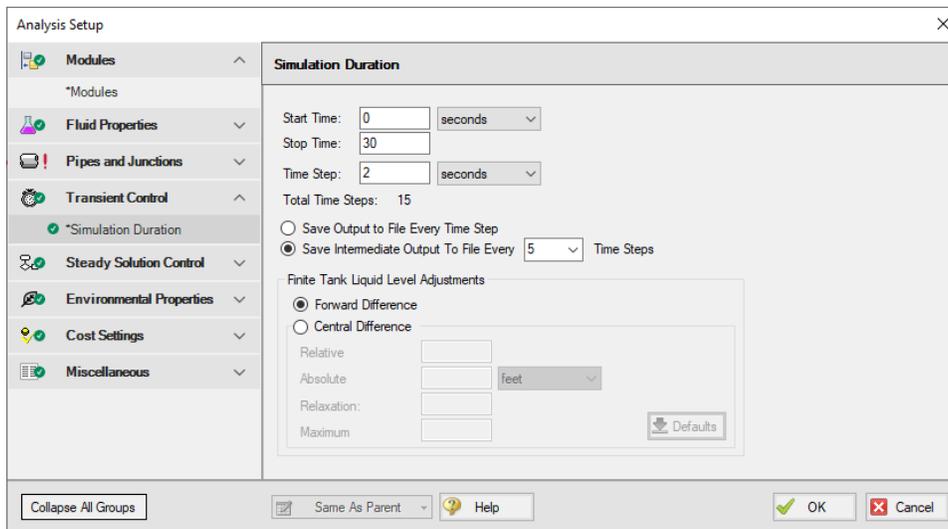


Figure 1: Simulation Duration panel fully defined with example values

The transient simulation is controlled by information the user enters in the Simulation Duration panel of the Transient Control group of the Analysis Setup window. This panel is used to specify the start and stop time of transient simulation, the size of the time step, and the frequency of the data saved to the transient output file.

Also available is [Forward Difference and Central Difference](#) options for Finite Tank Liquid Level Adjustments. If finite tank reservoirs are not used in the model, then there is no difference between the two options. However, there is an impact when one or more [finite tanks](#) do exist in the model.

Related Topics

XTS - Extended Time Simulation	585
Transient Control Group	143
Finite Tank Liquid Level Adjustments	145
Transient data did not extend to Stop Time at Junction X	792
System Life specified in Cost Settings is different from the XTS time	790

Related Examples

Beginner - Filling a Tank - XTS	1038
Reservoir Balance - XTS	1050
Valve Closing - XTS	1058
Housing Project - XTS	1072
Variable Demand - XTS	1081
Controlled Heat Exchanger Temperature - XTS	1066
Fixed Head Supply Tank - GSC XTS	1263

Finite Tank Liquid Level Adjustments

(XTS Module Only) The [Simulation Duration panel](#) allows one to set forward and central difference options. The forward and central difference options are only relevant when finite tanks are modeled. They relate to what liquid levels are used in the tank for each time step calculation. If the assumed time step is small enough, then the two methods yield virtually the same results. However, as the time step is increased, the central difference method is much more accurate. The drawback is that the central difference method is iterative and takes more computation time.

Forward Difference Method

The forward difference method solves the pipe system as if all tank levels remain at the same value as the beginning of the time step. Since the liquid level is known at the beginning of the time step, no time-based iterations are required. (The [iterations for flow and pressure](#) common to steady-state modeling still occur).

Once the solution is complete for the time step, the solved flowrates are used to update the tank liquid level for the next time step. Figure 1 depicts how this works.

Since in reality the tank level changes during the time step, precision is lost by using the liquid level at the beginning of the time step. As discussed earlier, making the time steps shorter helps alleviate this problem.

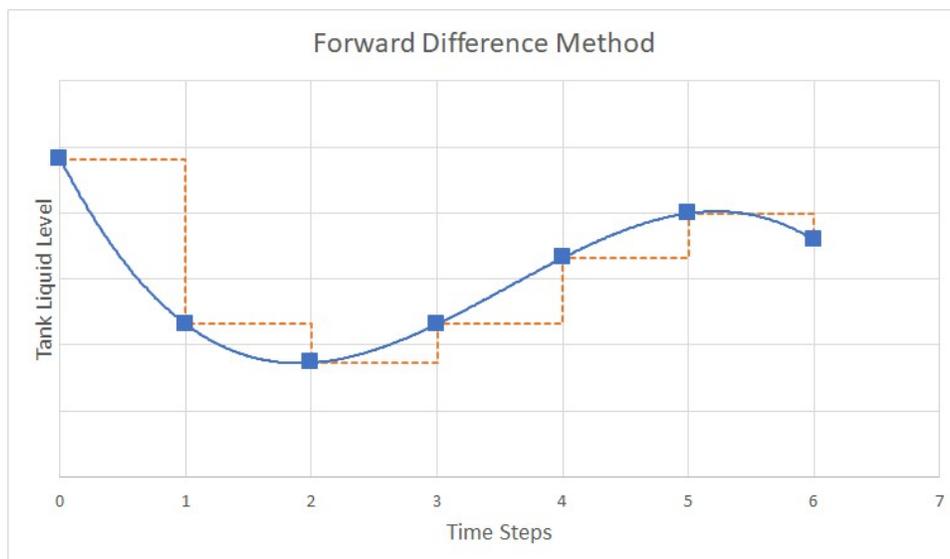


Figure 1: Forward difference method for time calculation assumes liquid level stays constant during each time step, allowing a direct calculation for each time

Central Difference Method

The central difference method solves the pipe system using tank liquid levels which represent the average over the time step (see Figure 2). Since the liquid level changes during the time step, it is more

accurate to use the average over the time step. However, the difficulty is that the average is taken between the liquid level at the beginning and end of the time step and, while the liquid level is known at the beginning, it is not known at the end. That is what we are trying to solve for. Hence the transient solution must assume a liquid level at the end of the time step and iterate on that value until convergence.

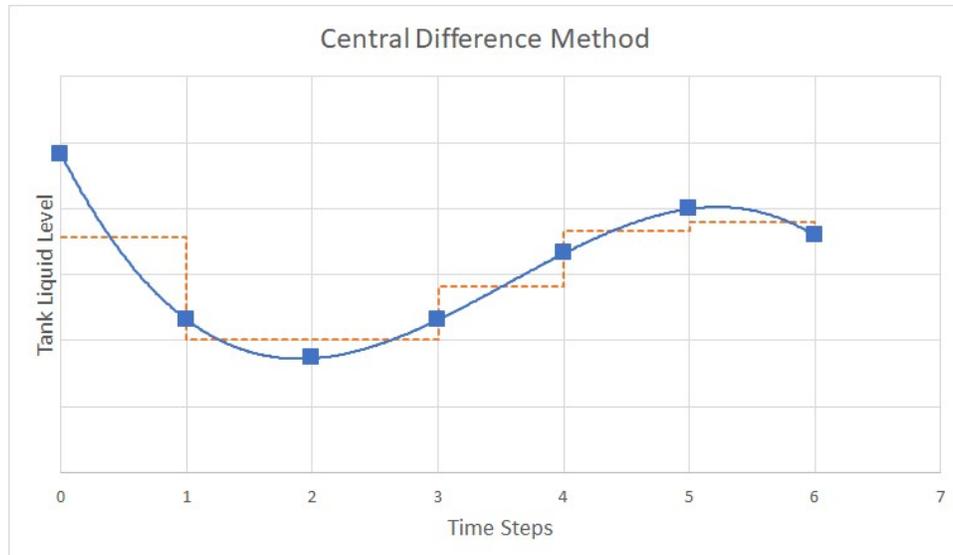


Figure 2: Central difference method for time calculation uses average liquid level during time step for calculation, requiring iteration for each time

There are several iteration control parameters needed by the central difference method, and these are input in the [Simulation Duration panel](#)(Figure 3). The central difference method attempts to converge on the liquid height at the next time step for each finite tank.

Relative and Absolute Tolerance

The relative and absolute tolerance inputs influence when AFT Fathom concludes it has converged on the new liquid height. Both of these criteria are applied, and if either one is satisfied then convergence has occurred.

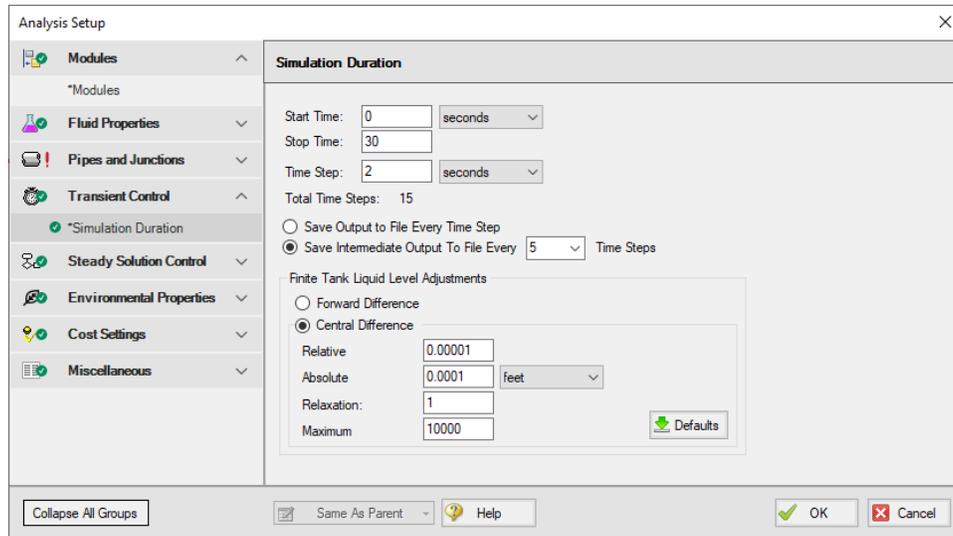


Figure 3: Iteration control parameters are required for the central difference method

The relative criteria is how much the liquid level calculation changes between iterations on a relative, or percentage basis. The absolute criteria relates how much the liquid level changes between iterations on an absolute basis of height. Hence the absolute tolerance has units of length associated with it.

The default values for relative tolerance is 0.00001, and the absolute tolerance is 0.0001 feet.

Relaxation

The relaxation applies to how fast AFT Fathom allows the liquid level to change during time steps. See the [Iterations panel](#) topic for a general discussion of relaxation.

Relaxation values must be greater than zero and less than or equal to 1. By default a value of 1 is used. This in fact means no relaxation.

Setting values closer to zero will improve convergence but slow the calculation. Values less than 0.01 should never be used unless directed by Applied Flow Technology.

Caution: The iteration control parameters for central difference time calculations should not be changed by the user unless they clearly understand their use or it has been recommended by Applied Flow Technology support. If changed indiscriminately, there is a danger of incorrect results being generated.

Maximum Iterations

This is the maximum number of attempts allowed for each time calculation iteration. If it reaches this value, the transient solution will stop due to failure to converge.

Changing Parameters During the Run

The Solution Progress window displays while the transient simulation is being performed. If a central difference method is used, because of its iterative nature the simulation can get stuck. If this happens then the simulation can be paused and Central Difference iteration parameters changed (such as reducing the relaxation parameter). With forward difference there are no parameters to change and hence this option is not available.

Output Values Displayed with Forward Difference

One advantage of the forward difference method is that it is easier to reconstruct the calculation for a given time step. For example, assume you want to run a separate steady-state AFT Fathom model to represent a particular time step calculation.

To do so, just look at the reservoir levels for that time step, enter these as the reservoir levels in the steady-state model, and all results should be the same as in the transient solution.

Output Values Displayed with Central Difference

When using the central difference method a somewhat strange result is that the reservoir liquid levels displayed in the output are never directly used to solve any time step (except for the initial “time zero” step). As discussed earlier, each time step is solved using an average liquid level. But this average liquid level is not displayed in the output. Instead, the liquid levels upon which the average was derived are displayed. This is depicted in Figure 3 where the liquid levels in the output do not correspond directly with the assumed liquid level for calculation. The squares represent the actual liquid level output values for central difference.

If for some reason you would like to construct a steady-state model to represent a time step from a transient simulation which used central difference, you must enter reservoir liquid levels which are the average of the desired time step and the previous time step.

Related Topics

Simulation Duration Panel..... 143

Goal Seek and Control Group

[\(GSC Module Only\)](#) The GSC module is controlled through the Goal Seek and Control items in the Analysis Setup window. In these panels, the user specifies all of the system variables, as well as the desired goals. As the user adds variables and goals to the model, AFT Fathom will display a symbol beside the pipes and junctions that have variables or goals applied to them. The default is a V for variables, and a G for goals. The goal symbol is not displayed next to objects that are part of a group goal.

The Goal Seek and Control Group has three panels:

- [Variables panel](#) - Allows users to define and apply variables to be used for the GSC run.
- [Goals panel](#) - Allows users to define and apply goals to be used for the GSC run.
- [Numerical Controls panel](#) - Allows GSC search engine parameters to be adjusted for troubleshooting purposes.

Related Topics

Analysis Setup Overview.....	83
Modules Group.....	84
Fluid Properties Group.....	90
Pipes and Junctions Group.....	141
Transient Control Group.....	143
Steady Solution Control Group.....	162
Environmental Properties Group.....	168
Cost Settings Group.....	170
Automatic Sizing Group.....	173
Miscellaneous Group.....	175
GSC - Goal Seek & Control.....	538
Variables Panel.....	149
Goals Panel.....	155
Numerical Controls Panel.....	161

Related Examples

Beginner - Heat Transfer in a Pipe - GSC.....	994
Spray Discharge System - GSC.....	1019
Pump Sizing and Selection with Flow Control Valves - GSC.....	1001
Cooling System - GSC.....	1006
Controlled Heat Exchanger Temperature - GSC.....	1027
Model Calibration with Pipe Variables - GSC.....	1011
NFPA Firewater System - GSC.....	1032
Fixed Head Supply Tank - GSC XTS.....	1263

Variables Panel

[\(GSC Module Only\)](#) The Variables panel is where variables are defined and applied for use with the GSC module (Figure 1). Variables are input parameters that the GSC module will change to achieve the user's desired goals. Generally, there must be one variable applied for each goal that is applied.

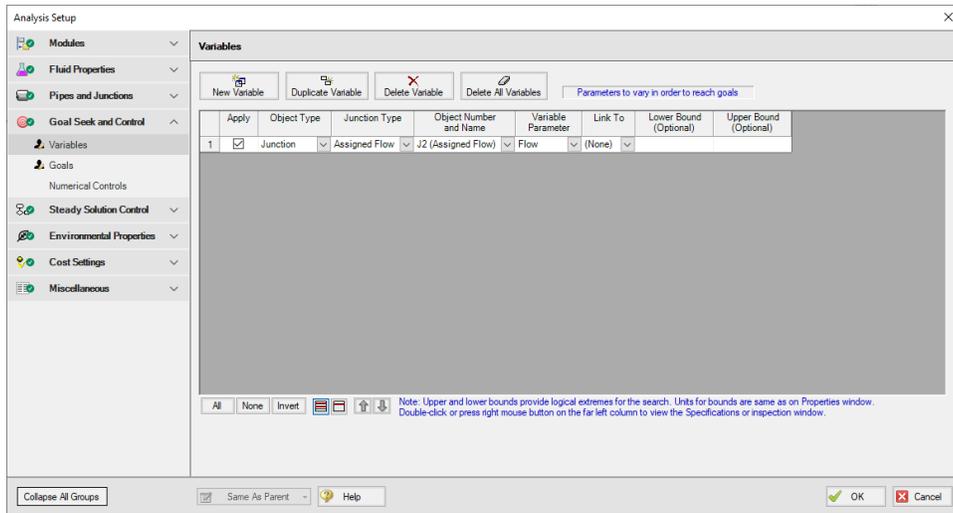


Figure 1: GSC Variables are input parameters to be automatically varied

Types of Variables

All variables are junction or pipe parameters. The variables that are available for each object type are shown in Table 1. There are over 30 types of variables.

Table 1: List of object types with available variables

Object Type	Variables	Restrictions
Pipes	Design Factor Friction ID Reduction (Scaling) Insulation Thickness	
Assigned Flow	Flow Elevation Temperature	Only with Heat Transfer
Assigned Pressure	Pressure Elevation Temperature	Only with Heat Transfer
Branch	Imposed Flow	
Check Valve	Loss Value	
Control Valve	Control Setpoint	
General Component	Loss Factor	
Heat Exchanger	Loss Factor Heat Rate Discharge Temperature Temperature Rise Enthalpy Rise Heat Transfer Area Overall Heat Transfer Coefficient Secondary Flow Secondary Inlet Temperature	Only with Heat Transfer Only with Heat Transfer

		Heat Transfer Only with Heat Transfer
Orifice	Diameter/Area Loss Factor Elevation	
Pump	Speed Flow Head Rise Impeller Size Control Setpoint	
Relief Valve	Loss Value	
Reservoir	Liquid Level Surface Pressure Temperature	Only with Heat Transfer
Screen	Open Area Loss Factor	
Spray Dis-charge	Spray Area K (Fire Sprinkler) Number Sprinklers Number Sparger Holes Elevation	
Three-Way Valve	Open Percentage	
Valve	Loss Value Elevation (Exit Valves)	Only for Exit Valves

Creating and Deleting Variables

To create a variable, click the New Variable button on the Variables panel (Figure 1). This will insert a new item into the variable list. After the variable has been added, the variable data is defined by entering

the necessary data in each of the data columns.

Variables may also be added by duplicating an existing variable, then modifying the data for the new variable. To duplicate a variable, select the variable to be duplicated from the variable list, and click the Duplicate Variable button.

To delete a variable, select the variable by clicking on the appropriate row in the variable list. After selecting the variable to be deleted, click the Delete Variable button. If you want to delete all of the variables in the list, click the Delete All Variables button.

Defining Variables

For each variable the following settings can be defined:

- **Object type** - The Object Type defines whether the variable applies to a pipe or a junction.
- **Junction type** - The Junction Type column is used to define the type of junction to which the variable is being applied.
- **Object number and name** - The specific junction to vary is selected in the Junction Number and Name column. The list will only display junctions of the type selected in the Junction Type column.
- **Variable parameter** - Each junction type has a specific set of parameters that are available as variables (Table 1). The list will only display parameters that are available for the junction type selected in the Junction Type column.
- **Linking variables** - The Link To column is used to force a variable on multiple junctions of the same type to solve to the same value.

An example where this would be desirable would be when using pump impeller trim as a variable for several pumps in parallel. It is often desirable to solve for a condition where the pump impeller trim is the same for all of the pumps. This can be accomplished by using variable linking. A pump impeller trim variable would be defined for each of the pumps. Then, one of the pumps would be selected as the basis pump, which just means no linking is specified for this pump. The other pumps would be linked to the basis pump by selecting the basis pump from the list in the Link To column.

- **Variable bounds** - During a goal search, the GSC module will modify the values for the defined variables. Sometimes it is helpful to specify upper and lower bounds for variables to provide logical extremes during the goal search. Some examples of logical bounds that can be applied would be lower and upper bounds on valve open percentage of 0% and 100%, respectively.

Note that bounds can have engineering units, but these units are not displayed. It is assumed the units are the same as the junction input parameter Properties Window.

Organizing Content

The lower left corner of the panel features options for rearranging and selecting variables and goals. All will select all available variable or goals. None will deselect all variables or goals. Invert will invert which variables and goals are selected. There are also options to show all variables or goals and to show only the applied variables or goals. The goals and variables can also be reordered using the turquoise arrows to change their position.

The Same As Parent button allows the variables, goals, numerical control, or all of the parameters (in regards to the Goal Seek and Control group) of the parent scenario to be copied down to the child scenario.

Reviewing Pipe/Junction Data

Data for the selected objects in the Variables panel can be reviewed and edited without having to close the Analysis Setup window.

- **Reviewing object input data** - If you would like to see the input data for the junction selected as a variable, press the right mouse button on the far left column in the Variables table. The [inspection window](#) will display, showing you the input data.
- **Editing object input data** - Similar to the inspection feature just described, you can open the [Pipe](#) or [Junction Properties window](#) by double-clicking the far left column in the Variables table. From there you can change any data desired.

Applying Variables

Once a variable has been created, the user must specify if a variable is to be used when the GSC module is run. To apply a variable, select the checkbox in the Apply column.

Using the Apply feature allows the user to define multiple variables that can be used in alternate cases or analyses. Any of the variables that are not being used for a particular analysis can remain in the list for later use. Click the Apply checkbox for the variables that are to be used in the current analysis.

Related Topics

GSC - Goal Seek & Control.....	538
Goal Seek and Control Group.....	148
Goals Panel.....	155
Numerical Controls Panel.....	161

Related Examples

Beginner - Heat Transfer in a Pipe - GSC.....	994
Spray Discharge System - GSC.....	1019
Pump Sizing and Selection with Flow Control Valves - GSC.....	1001
Cooling System - GSC.....	1006
Controlled Heat Exchanger Temperature - GSC.....	1027
Model Calibration with Pipe Variables - GSC.....	1011
NFPA Firewater System - GSC.....	1032
Fixed Head Supply Tank - GSC XTS.....	1263

Goals Panel

([GSC Module Only](#)) The Goals panel is where goals are defined and applied for use with the GSC module (Figure 1). Goals are output values you would like to achieve. The GSC module adjusts the applied variables until the applied goals are met. Generally there should be one goal applied for each applied variable.

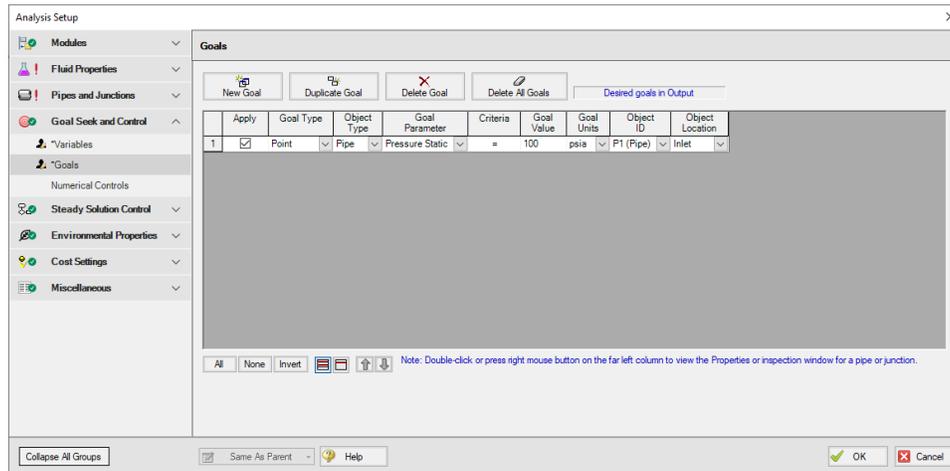


Figure 1: GSC Goals are output parameters to be achieved by changing the variables

Types of Goals

The GSC module offers over 150 goals applied to pipes, junctions or groups of pipes and/or junctions. Table 1 lists all of the types of goals.

Table 1: List of object types with available goals

Object Type	Goals	Restrictions/Comments
Pipe	Energy Gradeline Head Gradient Head Loss Heat Rate Into Pipe* Hydraulic Gradeline Mass Flow Rate Pressure Drop Friction Pressure Gradient Pressure Stagnation Pressure Static Pressure Static Maximum Pressure Static Minimum Temperature* Temperature Insulation #1 Surface* Temperature Insulation #2 Surface* Temperature Insulation #3 Surface* Temperature Loss* Temperature Outer Surface* Velocity Volumetric Flow Rate Wall Temperature*	*Only With Heat Transfer
Area Change	Energy Gradeline	
Bend	Hydraulic Gradeline	

Fundamental Modeling Concepts

Check Valve General Component Jet Pump Orifice Relief Valve Screen Valve Venturi Volume Balance	Mass Flow Rate Pressure Stagnation Pressure Static Temperature* Volumetric Flow Rate	*Only With Heat Transfer
Branch	Energy Gradeline Hydraulic Gradeline Mass Flow Rate* Pressure Stagnation Pressure Static Temperature** Volumetric Flow Rate*	*Net Flow Rate **Only With Heat Transfer
Assigned Flow	Energy Gradeline Hydraulic Gradeline Pressure Stagnation Pressure Static	
Assigned Pressure Reservoir	Mass Flow Rate Net into Jct* Vol. Flow Rate Net into Jct*	*Net Flow Rate
Control Valve	Cv* Energy Gradeline** Hydraulic Gradeline** Kv* Mass Flow Rate*** Open Percentage*	*If Open Percent Data Exists **Only for FCV ***Only for PRV, PSV ^Not PDCV ^^Only with Heat Transfer

Fundamental Modeling Concepts

	Pressure Loss [^] Pressure Stagnation ^{**} Pressure Static ^{**} Temperature ^{^^} Volumetric Flow Rate ^{***}	
Dead End	Pressure Hydraulic Gradeline	
Heat Exchanger	Energy Gradeline Heat Rate In* Hydraulic Gradeline Mass Flow Rate Pressure Stagnation Pressure Static Temperature ^{**} Volumetric Flow Rate	*Only with Heat Transfer. For non-fixed heat rate exchanger models. **Only with Heat Transfer. For non-controlled temperature models.
Pump	% of BEP Energy Gradeline Head Rise* Hydraulic Gradeline Mass Flow Rate ^{**} NPSH Margin Absolute NPSH Ratio Power Overall Pressure Rise* Pressure Stagnation Pressure Static	*Only for non-fixed head rise pumps **Only for non-fixed flow pumps

	Temperature*** Volumetric Flow Rate**	***Only with Heat Transfer
Spray Discharge	Mass Flow Rate Out* Volumetric Flow Rate Out*	*Net Flow Rate
Tee or Wye Three Way Valve	Energy Gradeline Hydraulic Gradeline Pressure Stagnation Pressure Static Temperature*	*Only With Heat Transfer

Creating and Deleting Goals

To create a goal, click the New Goal button on the Goals tab. This will insert a new item into the goals list. After the goal has been added, the goal data is defined by entering the necessary data in each of the data columns.

Goals may also be added by duplicating an existing goal, then modifying the data for the new goal. To duplicate a goal, select the goal to be duplicated from the goal list, and click the Duplicate Goal button.

To delete a goal, select the goal by clicking on the appropriate row in the goal list. After selecting the goal to be deleted, click the Delete Goal button. If you want to delete all of the goals in the list, click the Delete All Goals button.

Defining Goals

For each goal the following settings can be defined:

- **Goal Type** - Select the type of goal to be used from the following:
 - **Point Goal** - A point goal is specified when the desired goal can be specified at a specific object location in the model.
 - **Differential Goal** - A differential goal is used when the goal value is actually determined by the difference in values between two specific object locations. An example of a differential goal might be the pressure difference between two locations in the model.
 - **Group Goal** - A Group goal is used when a goal is to be applied to a group of objects at the same time. In order to use a Group goal, all of the objects must be added to a [Group](#) by using the Groups command on the Edit menu.
- **Object Type** - Select the type of object to which the goal is to be applied. See the section below for further description of the object types.

- **Goal Parameter** - Select the type of goal to be applied to the selected object, such as pressure, flow rate, or temperature.
- **Criteria** - Specify the criteria which is to be met for the specified goal parameter, such as equals, greater than or equal, or less than or equal.
- **Goal Value/Units** - Specify the value and units for the goal parameter.
- **Object ID** - Specify the pipe, junction, or group name to which the goal applies.
- **Object Location** - Specify, as appropriate, whether the goal applies to the inlet or outlet of the selected object or group.

Organizing Content

The lower left corner of the panel features options for rearranging and selecting variables and goals. All will select all available variable or goals. None will deselect all variables or goals. Invert will invert which variables and goals are selected. There are also options to show all variables or goals and to show only the applied variables or goals. The goals and variables can also be reordered using the turquoise arrows to change their position.

The Same As Parent button allows the variables, goals, numerical control, or all of the parameters (in regards to the Goal Seek and Control group) of the parent scenario to be copied down to the child scenario.

Object Types

For **Point and Differential goal types**, the object type indicates whether the goal is to be applied to a pipe or a junction object. If the goal is to be applied to a junction, the type of junction is selected from the list.

For the Group goal type, the user must select between a Group Max/Min or a Group Sum object type.

A **Group Max/Min goal** allows a single goal to be applied to a group of objects. The GSC module applies a Group Max/Min goal by ensuring the final goal value is either greater than or equal to (a Min goal) or less than or equal to (a Max goal) the specified value for each object in the group. For example, a Group Max/Min goal can be applied to ensure a minimum flow rate out of each spray discharge nozzle in a group of nozzles. Group Max/Min goals only work for groups where all group members are of the same type (i.e., all pipes or all valve junctions).

A **Group Sum goal** is used to set the sum of the goal parameter for all the members in a group to a specified value. For example, the Group Sum goal can be applied to set the sum of the volumetric flow rates through all of the nozzles in a group of spray discharge nozzles to a particular value. Groups used in Group Sum goals can consist of pipes and/or junctions, with junctions of different type.

Applying Goals

Once a goal has been created, the user must specify if a goal is to be used when GSC is run. To apply a goal, select the checkbox in the Apply column.

Using the Apply feature allows the user to define multiple goals that can be used in alternate cases or analyses. Any of the goals that are not being used for a particular analysis can remain in the list for later use. Click the Apply checkbox for the goals that are to be used in the current analysis.

Reviewing Pipe/Junction Data

Data for the selected objects in the Goals panel can be reviewed and edited without having to close the Analysis Setup window.

- **Reviewing object input data** - If you would like to see the input data for the junction selected as a goal, press the right mouse button on the far left column in the Goals table. The [inspection window](#) will display, showing you the input data. This feature does not work for group goals.
- **Editing object input data** - Similar to the inspection feature just described, you can open the [Pipe](#) or [Junction Properties Window](#) by double-clicking the far left column in the Goals table. From there you can change any data desired. This feature does not work for group goals.

Related Topics

GSC - Goal Seek & Control.....	538
Goal Seek and Control Group.....	148
Variables Panel.....	149
Numerical Controls Panel.....	161
When Goals Cannot Be Achieved.....	545
GSC Goal may not be valid. User goal = X. Final goal result = Y.....	778

Related Examples

Beginner - Heat Transfer in a Pipe - GSC.....	994
Spray Discharge System - GSC.....	1019
Pump Sizing and Selection with Flow Control Valves - GSC.....	1001
Cooling System - GSC.....	1006
Controlled Heat Exchanger Temperature - GSC.....	1027
Model Calibration with Pipe Variables - GSC.....	1011
NFPA Firewater System - GSC.....	1032
Fixed Head Supply Tank - GSC XTS.....	1263

Numerical Controls Panel

[\(GSC Module Only\)](#)

Numerical control of the GSC search engine parameters can be adjusted on the Numerical Control tab. It is recommended the user **avoid** changing these parameters unless the user understands their purpose or it has been recommended by AFT technical support.

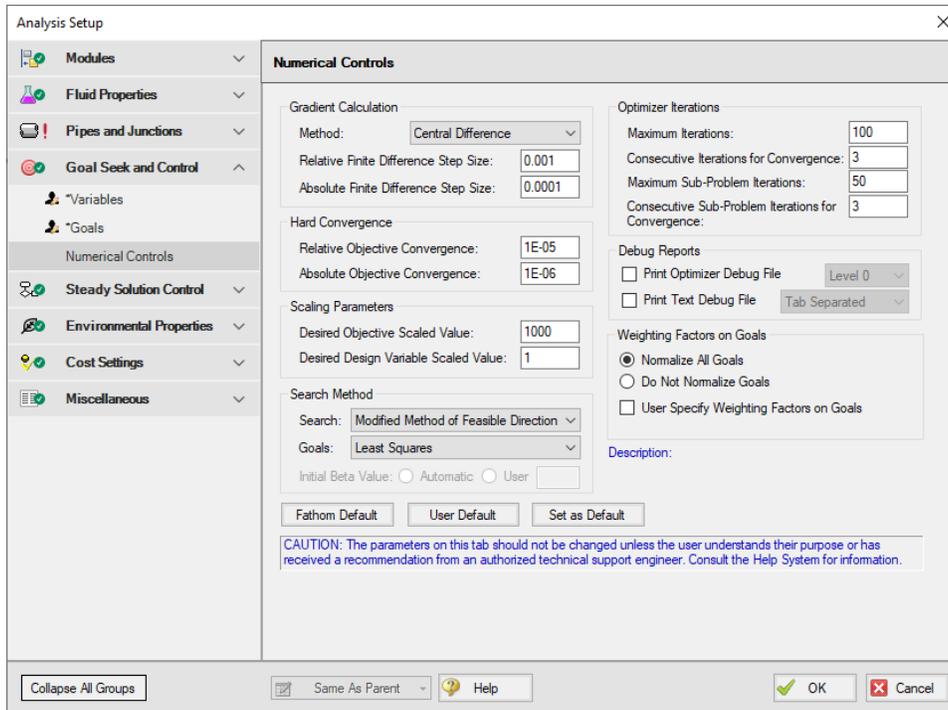


Figure 1: Default Numerical Controls panel setup in the Goal Seek and Controls Group

Related Topics

GSC - Goal Seek & Control.....	538
Goal Seek and Control Group.....	148
Variables Panel.....	149
Goals Panel.....	155

Steady Solution Control Group

The user has control over a number of parameters that influence numerical convergence.

Note: These values should only be changed by experienced users or under advisement by the AFT Support Team. **Poorly defined Solution Control settings can cause the model to converge on an incorrect solution without warning messages.**

- [Tolerance panel](#) - The iterative tolerances that define convergence can be defined on this panel.
- [Iterations panel](#) - Additional iteration controls that can affect whether or not a model converges, or how quickly it will converge, can be defined here. Most settings on this panel will not affect the final result.
- [Temperature Limits panel](#) - Options to limit the temperature results in the Solver when [heat transfer is enabled](#).

Related Topics

Analysis Setup Overview.....	83
Modules Group.....	84
Fluid Properties Group.....	90
Pipes and Junctions Group.....	141
Transient Control Group.....	143
Goal Seek and Control Group.....	148
Environmental Properties Group.....	168
Cost Settings Group.....	170
Automatic Sizing Group.....	173
Miscellaneous Group.....	175
Tolerance Panel.....	163
Iterations Panel.....	165
Temperature Limits Panel.....	167

Tolerance Panel

The iterative tolerances that define convergence are defined on the Tolerance panel in [Analysis Setup](#), shown in Figure 1. These values can be modified for Pressure Tolerance, Flow Rate Tolerance, and Temperature Tolerance (when Heat Transfer is enabled).

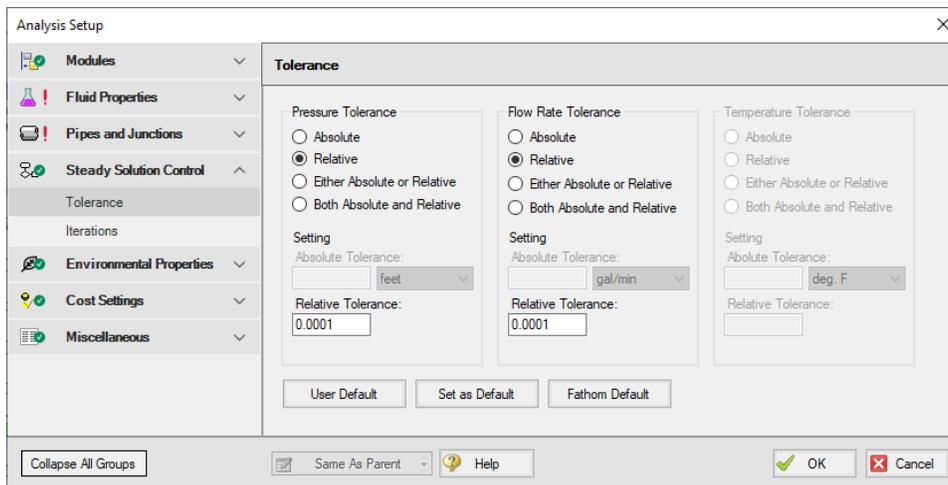


Figure 1: Tolerance panel in Analysis Setup

An iterative approach is used to obtain pipe flow solutions. When approaching any analysis iteratively, a stopping point must be defined. The simplest approach is to stop after a certain number of iterations. However, this either causes uncertain results because the result is still changing, or wasted processing time because there are too many iterations.

A solution is to use **iterative tolerance**. This tolerance is related to the change between iterations. When this change is smaller than the iterative tolerance, the solution is said to be *converged*.

Tolerance and Solution Accuracy

Iterative tolerance does not have any relationship to an **engineering tolerance**.

An **engineering tolerance** is a familiar term to most engineers - it is a permissible limit in variation of a parameter. A simple example is the length of a steel bar - an engineer might specify that this length must be 3 ± 0.01 meters. A steel bar manufactured by this specification is *known* to be within the engineering tolerance of the "true" or desired length. In other words, **engineering tolerance** describes an *accuracy*.

Iterative tolerances do *not* directly describe an accuracy. Convergence *only* means that the change between iterations was within the specified limit.

With sound iterative methods, and an appropriately specified iterative tolerance, the iterative solution will be close to the true solution. However, even with good settings, the iterative tolerance does not define **how close** the results are to the true solution.

If there is concern that the results may not be close enough to the true answer, the tolerances could be tightened and the results compared. If the results do not change appreciably, an appropriate tolerance was likely selected.

Avoiding False Convergence

It is important to keep the tolerance at least one order of magnitude smaller (two orders or more is recommended) than the [relaxation](#). A false convergence can occur when the change from the old value to the new value is small enough to be within the tolerance because it is restricted by the relaxation.

Absolute Tolerance

Absolute tolerance looks at the *absolute* change in quantity of a given parameter. If the change is smaller than the directly specified value, the parameter is considered converged.

This approach works equally well for both small and very large values, making iterative solvers more likely to find convergence. However, specifying an appropriate absolute tolerance requires some knowledge of the true solution. The tolerance should be set 3 or 4 orders of magnitude below the solution, if there are near zero flows in the system, this will require a very low tolerance.

Relative Tolerance

Relative tolerance, on the other hand, checks the *relative* change of a given parameter. Each change is divided by the number itself, giving a change proportional to the magnitude.

This approach does not require any knowledge about the true solution, and for that reason is the default method.

Table 1 shows the difference between relative and absolute change while iterating. Notice that the absolute change has units, while relative change does not.

Convergence criteria can also be based on either (or both) absolute or relative change. Table 1 shows how this is applied.

Table 1: Example of four tolerance methods for a mass flow rate example. Absolute tolerance = 0.0005 lbm/s, Relative tolerance = 0.0001

Iteration #	Mass Flow Rate (lbm/sec)	Relative Change	Absolute Change (lbm/sec)	Meets Abs. Tolerance?	Meets Rel. Tolerance?	Meets Abs. OR Rel. Tolerance?	Meets Abs. AND Rel. Tolerance?
1	20.0000	-	-	-	-	-	-
2	12.0000	0.666667	8.000000	NO	NO	NO	NO
3	14.0000	0.142857	2.000000	NO	NO	NO	NO
4	13.5000	0.037037	0.500000	NO	NO	NO	NO
5	13.6000	0.007353	0.100000	NO	NO	NO	NO
6	13.5900	0.000736	0.010000	NO	NO	NO	NO
7	13.5913	0.000096	0.000130	NO	YES	YES	NO
8	13.5907	0.000044	0.000600	NO	YES	YES	NO
9	13.5909	0.000015	0.000200	YES	YES	YES	YES

Related Topics

Steady Solution Control Group.....	162
Iterations Panel.....	165
Temperature Limits Panel.....	167
Try Absolute Tolerance.....	804
Setting relaxation lower than relative pressure/flow tolerance gives unreliable results.....	774

Iterations Panel

Relaxation, iteration limits, and matrix method settings are defined on the Iterations panel in [Analysis Setup](#). Most settings on this panel will not affect the final result. The default state of the Iterations panel is shown in Figure 1.

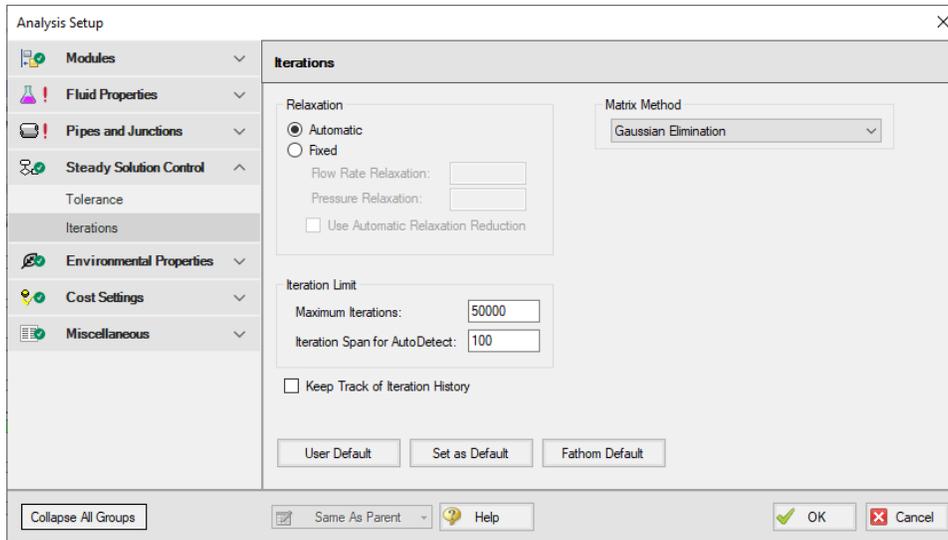


Figure 1: Iterations panel in Analysis Setup

Relaxation

Relaxation slows iterative progress toward a solution reducing the destabilizing effect of non-linear systems.

- **Automatic** - Implements an adaptive relaxation scheme that reduces the relaxation dynamically in response to highly non-linear features of the problem.
- **Fixed** - Allows for user specified Flow Rate and Pressure Relaxation

When applied to the flow solution, relaxation r is used as follows:

$$\dot{m}_{new} = r(\dot{m}_{new} - \dot{m}_{old}) + \dot{m}_{old}$$

A relaxation of 1 is the same as no relaxation. Each iteration's prediction is used directly in the next iteration.

A relaxation of 0 is not allowed, as this would prevent the parameter from changing.

- **Flow Rate Relaxation** - For highly non-linear systems, a flow rate relaxation of 0.1 is advised. Experience shows that using flow rate relaxation factors between 0.5 and 1 has limited usefulness. If a flow rate relaxation of 1 does not work, you should generally try flow rate relaxation parameters between 0.1 and 0.5. In more extreme cases, a flow rate relaxation factor smaller than 0.1 may be required.
- **Pressure Relaxation** - Because of the nature of the matrix solver, experience indicates that if pressure relaxation is used, it should always be set to 0.5 or 1 to maintain stability.

Iteration Limit

- **Maximum Iterations** - 50000 by default, sufficient for the vast majority of models. Most models converge in a relatively small number of iterations. This limit does not directly affect the solution.
- **Iteration Span for AutoDetect** - 100 by default, if the Solver detects that no meaningful progress has been made to convergence within this many iterations, relaxation is automatically lowered. This also automatically adjusts [tolerance](#).

Matrix Method

[Network analysis](#) requires solving complex matrices potentially thousands of times. Different methods may solve faster or slower, but have no bearing on results. The default method of Gaussian Elimination is highly robust and usually provides the fastest convergence. The available Matrix Methods are:

- Gaussian Elimination
- Gaussian Elimination With Pivoting
- LU Decomposition

Iteration History

Iteration History shows a history of which pipe or junction is most [out of tolerance](#), which is helpful for troubleshooting. After enabling Iteration History on the Iterations panel in [Analysis Setup](#), the Iteration History can be viewed from the [Solution Progress window](#) by pausing the calculation and selecting the Other Actions button.

Related Topics

Steady Solution Control Group.....	162
Tolerance Panel.....	163
Temperature Limits Panel.....	167
Lower the Flow Rate Relaxation.....	803
Review the Iteration History.....	804
Setting relaxation lower than relative pressure/flow tolerance gives unreliable results.....	774
Solution Progress Window.....	383

Temperature Limits Panel

The Temperature Solver Limits panel allows users to set boundaries for temperature within the solver. These limits are intended to help with convergence and prevent the solver from providing unrealistic results. These limits are intended to keep the solver within physical restraints, such as temperatures that can't be produced on Earth. This is most useful when operating near the edge of available physical property data. During iteration, the Solver may attempt to extrapolate past the available data, returning an error. Limiting the allowable range can keep the search within the proper bounds.

The Temperature Limits Panel is available if [heat transfer](#) is enabled.

For design requirements and limitations, use [Design Alerts](#) to signify when the solvers values are beyond a desired range yet is still physically possible to occur.

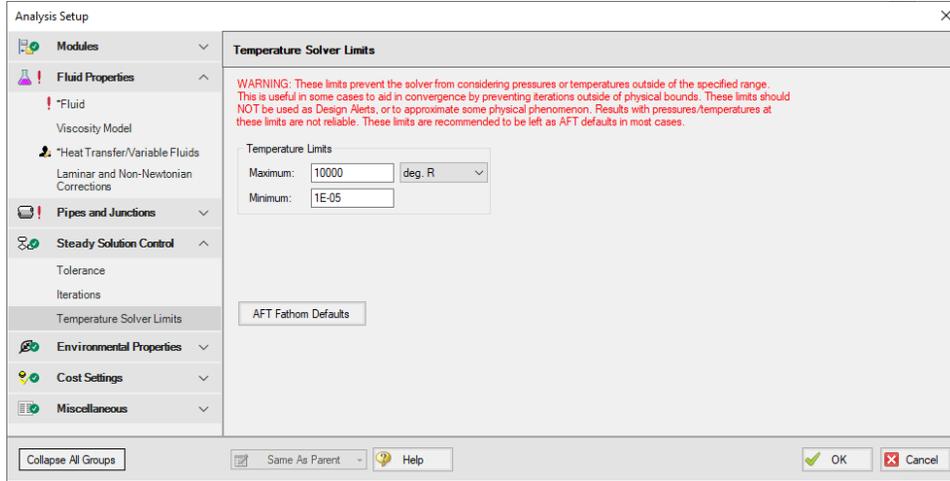


Figure 1: Temperature Limits panel in the Analysis Setup window

Related Topics

Steady Solution Control Group.....	162
Tolerance Panel.....	163
Iterations Panel.....	165
Pipe X temperature is equal to or greater/less than the maximum/minimum temperature limit of.....	780

Environmental Properties Group

The Environmental Properties Group in [Analysis Setup](#) has one item: [Environmental Properties panel](#). By default, the Environmental Properties Group is fully defined and will have a green checkmark next to its name.

Related Topics

Analysis Setup Overview.....	83
Modules Group.....	84
Fluid Properties Group.....	90
Pipes and Junctions Group.....	141
Transient Control Group.....	143
Goal Seek and Control Group.....	148
Steady Solution Control Group.....	162
Cost Settings Group.....	170

Automatic Sizing Group..... 173
 Miscellaneous Group..... 175
 Environmental Properties Panel..... 169

Environmental Properties Panel

The Environmental Properties panel, shown in Figure 1, controls the definition of the atmospheric pressure, gravitational acceleration, and transition Reynolds number used to model the system.

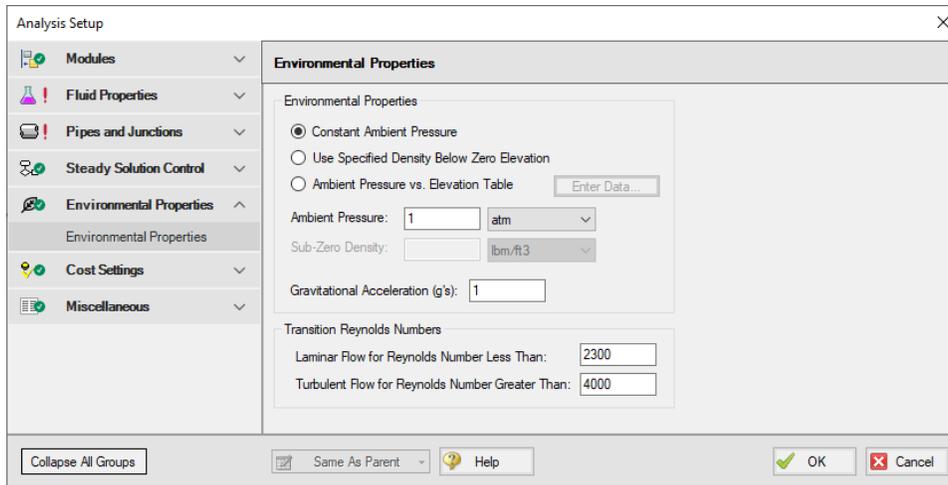


Figure 1: Default state of the Environmental Properties panel

Environmental Properties

Specify the ambient pressure and gravitational acceleration values for the solver to use in the calculations. By default, these values are 1 atm and 1 g, respectively.

Ambient Pressure

AFT Fathom uses a default atmospheric pressure of 1 standard earth atmosphere, or 14.696 psia. This pressure is also used to convert absolute pressures to gauge pressures. Keep in mind that atmospheric pressure varies with altitude. A project at Sea Level should not have the same atmospheric pressure as a project at 8500 feet. Fathom can automatically adjust atmospheric pressure using the Ambient Pressure vs. Elevation Table found in the [Analysis Setup window](#). Here, users can input a pressure profile that will change the ambient pressure depending on a junction's elevation.

Gravitational Acceleration

The default for gravitational acceleration is 1 standard earth gravitational acceleration, or 32.174 feet/sec². You can change this to a multiple of standard earth accelerations (that is, number of g's) according to the design environment for your system. In principle, the body force on a fluid system does not have to be due to gravity.

Transition Reynolds Number

AFT Fathom allows you to modify the transition Reynolds numbers. The default values assume that a Reynolds number of less than 2300 is laminar and a Reynolds number of greater than 4000 is turbulent. In the transition zone, a linear interpolation is assumed. This applies for friction factors and convection heat transfer.

You can modify these transition Reynolds numbers if you choose by manually typing in the values on the Environmental Properties panel in the [Analysis Setup window](#).

Related Topics

Environmental Properties Group.....	168
Head rise across fixed flow pumps is referenced to system g level.....	786
Head loss, head rise, hydraulic gradeline and energy gradeline output does not have meaning for zero g systems.....	786

Cost Settings Group

The Cost Settings Group in [Analysis Setup](#) has one item: the [Cost Settings panel](#). This group is always present regardless of whether or not cost calculation is enabled. When all fields are defined, the Cost Settings Group will have a green checkmark next to its name.

Related Topics

Analysis Setup Overview.....	83
Modules Group.....	84
Fluid Properties Group.....	90
Pipes and Junctions Group.....	141
Transient Control Group.....	143
Goal Seek and Control Group.....	148
Steady Solution Control Group.....	162
Environmental Properties Group.....	168
Automatic Sizing Group.....	173
Miscellaneous Group.....	175
Cost Settings Panel.....	171
Cost Calculation.....	516

Related Examples

Beginner - Cost Calculation.....	908
Plant Cooling Cost Calculation.....	958

Cost Settings Panel

Note: This topic does not apply for the [ANS module](#). To set up cost calculations with ANS, see the [Assign Cost Libraries](#) topic.

The Cost Settings panel must be defined in order to run a model. By default, cost calculation is not enabled and no further input is required to define the panel.

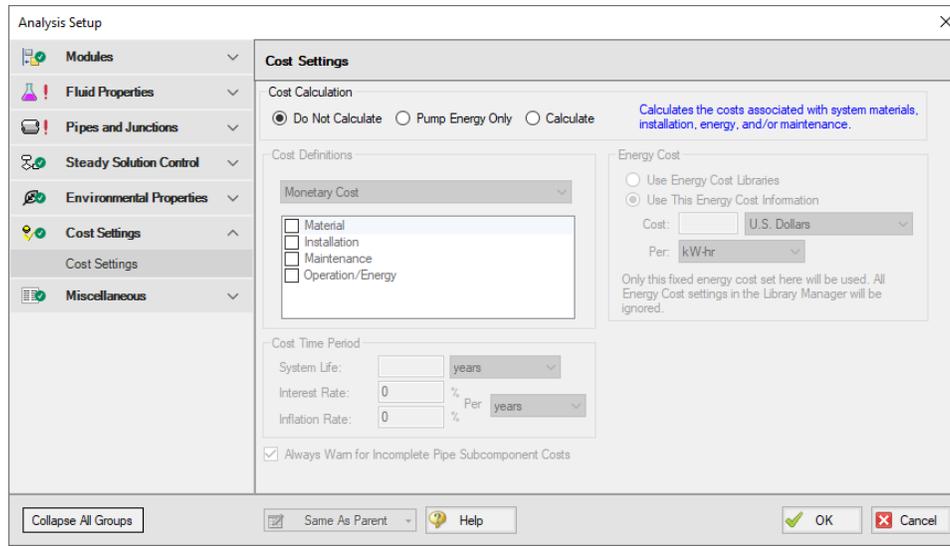


Figure 1: The Cost Settings panel

Enable Cost Calculation

The first step in any cost calculation is to enable it in the [Cost Settings panel](#). The following options are available:

- **Do not calculate** - Selected by default.
- **Pump Energy Only** - Only calculate the costs associated with the power consumption of pumps. The energy cost and system life must be entered in [Cost Settings](#).
- **Calculate** - Enable full calculation of costs. In addition to defining Cost Settings, [Cost Libraries](#) must be defined to use this feature.

Including Objects in Cost Report

Workspace objects must be included in the Cost Report to have their costs calculated. This allows only analyzing the cost of certain parts of the model - useful if only one pump is being replaced, or if an expansion to an existing system is being considered. To include the objects in the Cost Report simply select "Include Cost in Report" on the object's Cost tab, as shown below. Finer detail in how the Cost is included can be set by clicking the [Cost Application](#) button.

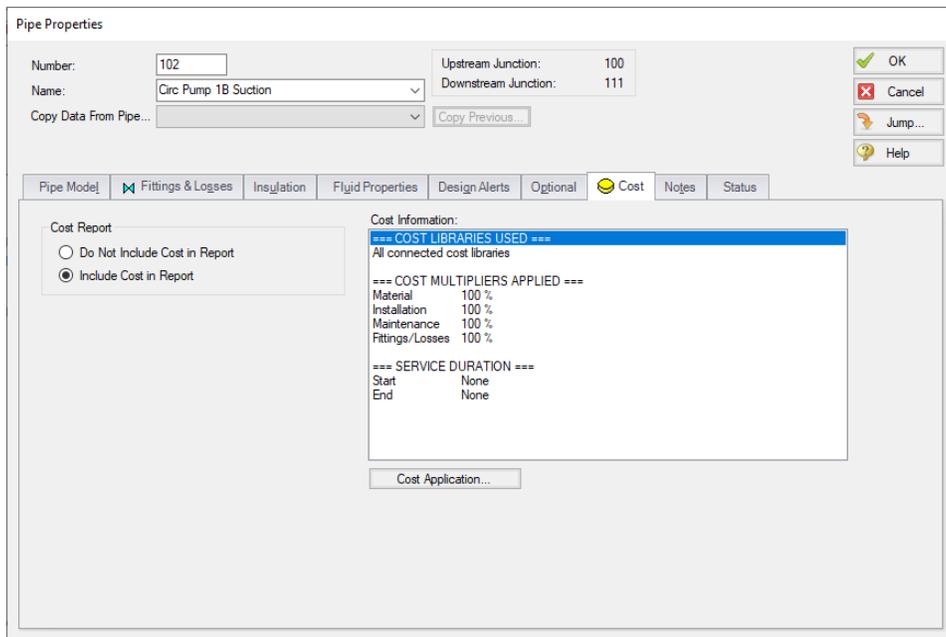


Figure 1: Include Cost in Report

Energy Cost

Energy cost data can be specified through an [Energy Cost Library](#), or by directly entering an energy cost on the Cost Settings panel. Using an energy cost library allows you to specify multiple energy costs, such as energy costs at both peak and off-peak rates.

Cost Definitions

The Cost Definitions section allows you to specify the type of costs to be determined. Check the boxes next to the items that are of interest. The options are:

- Material
- Installation
- Maintenance
- Operation/Energy

Cost Time Period

When Cost calculations are enabled the System Life is required. Interest Rate and inflation rate will default to zero, and may be changed by the user.

The System Life affects all recurring costs. It can also affect non-recurring costs if the Service Duration is used.

See [Cost Libraries](#) for detailed discussions regarding recurring and non-recurring costs.

Related Topics

Cost Settings Group.....	170
Cost Calculation.....	516
System Life specified in Cost Settings is different from the XTS time.....	790

Related Examples

Beginner - Cost Calculation.....	908
Plant Cooling Cost Calculation.....	958

Automatic Sizing Group

[\(ANS Module Only\)](#) The Automatic Sizing Group in [Analysis Setup](#) only appears when the ANS Module is activated from the [Modules panel](#). The Automatic Sizing Group has one panel: the [Sizing Summary panel](#).

Related Topics

Analysis Setup Overview.....	83
Modules Group.....	84
Fluid Properties Group.....	90
Pipes and Junctions Group.....	141
Transient Control Group.....	143
Goal Seek and Control Group.....	148
Steady Solution Control Group.....	162
Environmental Properties Group.....	168
Cost Settings Group.....	170
Miscellaneous Group.....	175
ANS - Automated Network Sizing.....	620
Sizing Summary Panel.....	174

Related Examples

Beginner - Three Reservoir Problem - ANS.....	1134
Control Valve - ANS.....	1149
Cooling System - Creating Libraries - ANS.....	1157
Cooling System - ANS.....	1176

Cooling System, Multiple Design Cases - ANS	1207
Hot Water System - ANS	1217
Housing Project - ANS	1246
Variable Drive - ANS	1231

Sizing Summary Panel

The Sizing Summary panel in [Analysis Setup](#) displays the [Sizing Summary](#) when the [ANS module](#) is active. The Sizing Summary can also be viewed in the [Sizing Window](#).

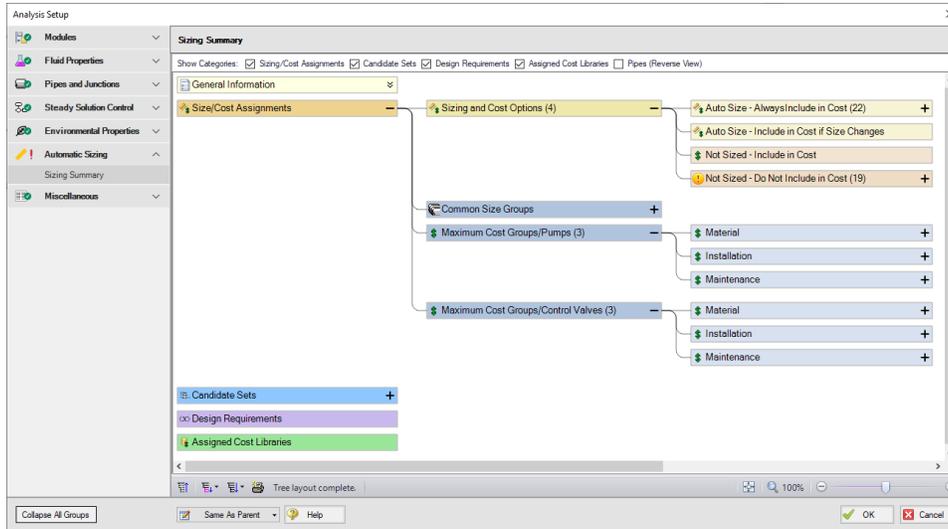


Figure 1: Sizing Summary panel with all tree nodes expanded

Related Topics

ANS - Automated Network Sizing	620
Automatic Sizing Group	173

Related Examples

Beginner - Three Reservoir Problem - ANS	1134
Control Valve - ANS	1149
Cooling System - Creating Libraries - ANS	1157
Cooling System - ANS	1176
Cooling System, Multiple Design Cases - ANS	1207
Hot Water System - ANS	1217
Housing Project - ANS	1246
Variable Drive - ANS	1231

Miscellaneous Group

The Miscellaneous group contains settings that are used infrequently. By default, the Miscellaneous Group is fully defined, with a green checkmark next to its name, without any additional input from the user.

The Miscellaneous group contains the following panels:

- [NFPA panel](#) - Allows NFPA to be activated to generate NFPA reports
- [Heat Transfer Sectioning panel](#) - Contains advanced settings for heat transfer calculations for troubleshooting purposes
- [Advanced Parameters panel](#) - Contains advanced parameters that can be used for troubleshooting purposes

Related Topics

Analysis Setup Overview.....	83
Modules Group.....	84
Fluid Properties Group.....	90
Pipes and Junctions Group.....	141
Transient Control Group.....	143
Goal Seek and Control Group.....	148
Steady Solution Control Group.....	162
Environmental Properties Group.....	168
Cost Settings Group.....	170
Automatic Sizing Group.....	173
Heat Transfer Sectioning Panel.....	178
Advanced Parameters Panel.....	179
NFPA Panel.....	175

NFPA Panel

The NFPA panel allows the user to enable NFPA settings, set the [Hazen Williams factor](#), and define an ambient pressure for NFPA calculations.

NFPA stands for the National Fire Protection Association. This association develops codes and standards that are used by engineers designing safety related systems related to fire protection. AFT Fathom provides users the ability to specify remote paths and automatically calculate the hydraulics through that path as is recommended in NFPA 15 Section 8.5.3.1 ([NFPA 15 \(2012\), pg 23](#)). When coupled with the GSC module (Goal Seek and Control), AFT Fathom includes the ability to [determine the hydraulically most remote path automatically](#).

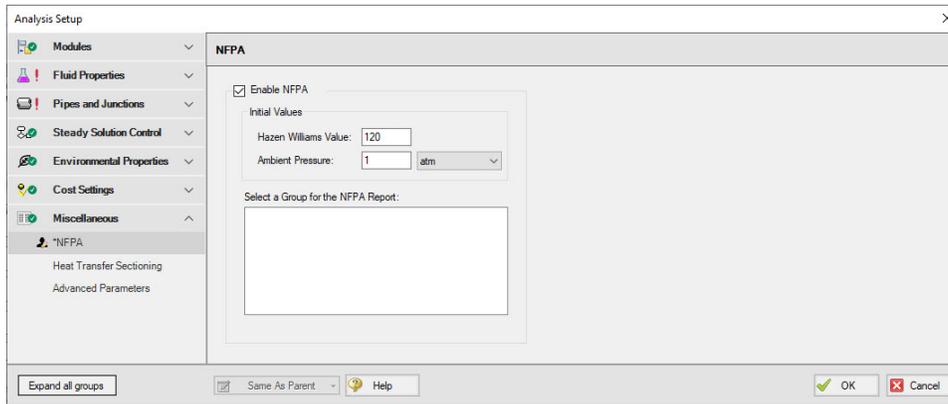


Figure 1: NFWA panel

NFWA Report

In order to generate the NFWA report, a group must be created using the [Group Manger](#). This group must contain at least one spray discharge. It is best practice for this group to be a single continuous flow path connecting the most hydraulically remote spray discharge to the pressure source for the system. Once the group is created, a user will need to select the group on the NFWA panel above. Multiple groups can be created, but only one can be selected per run. To see how this is done, see the [NFWA Fire-water System Example](#). For more detail on NFWA reports consult the NFWA standard.

Terminology

The table below shows equivalent naming conventions between Fathom and the NFWA standard. Stagnation (total) pressure is equal to static (normal) pressure plus dynamic (velocity) pressure.

AFT Fathom	NFWA Standard
Stagnation Pressure (P_o)	Total Pressure (P_t)
Static Pressure (P)	Normal Pressure (P_n)
Dynamic Pressure	Velocity Pressure (P_v)

Differences Between Fathom and NFWA Standards

The NFWA standard was built as a method that engineers could use to solve complex fire water networks via hand calculations. To accomplish this the NFWA standard uses certain assumptions which differ from the computational methods and precise solution methodology that Fathom uses in its Solver. Therefore, it is expected that Fathom and NFWA hand calculation results may differ. Some key differences are listed below:

- **Matrix Methods for Flow Splits** - Fathom uses a matrix of equations to calculate the exact flow splits, whereas the NFPA Standard uses conservative estimates for flow splits.
- **Velocity Changes at Flow Splits** - Fathom accounts for velocity changes at flow splits due to diameter changes and mass leaving the system at other nozzles.
- **Stagnation Pressure at Nozzle Exits** - Fathom uses stagnation pressure as the driving force for flow out of all nozzles (inline and end nozzles), whereas the NFPA standard uses static pressure for inline nozzles (total pressure minus velocity pressure). Therefore, Fathom does not adhere to NFPA 15 (2012) Section A.8.5.1.2 (pg 48).

Related Topics

Equivalent Length.....	267
Hazen-Williams NFPA.....	177
Miscellaneous Group.....	175
Heat Transfer Sectioning Panel.....	178
Advanced Parameters Panel.....	179

Related Examples

NFPA Firewater System.....	934
NFPA Firewater System - GSC.....	1032

Hazen-Williams NFPA

Some in industry use an alternate form of the Hazen-Williams formula which has this form (NFPA 13, 1999):

$$p = \frac{4.52 Q^{1.85}}{C^{1.85} d^{4.85}} \quad (1)$$

where:

- p = Pressure drop gradient (psi per foot)
- C = friction loss coefficient
- Q = Volumetric flow rate (gpm)
- d = Internal diameter (inches)

This differs slightly from the [Hazen Willams](#) version used in AFT Fathom.

While both [Equation 6](#) (and its variant [Equation 7](#)) in topic [Pressure Drop in Pipes](#) and Equation 1 (in this topic) are used heavily in industry, surprisingly they are not equivalent. The reason relates to Equation 1 being in terms of pressure drop but not having a fluid density in the equation. Equation 1 is intended for

water, but it says that the pressure drop is independent of fluid density. This is contrary to Equation 2 and almost all other references.

The definition of pressure loss as it relates to head loss is

$$\Delta P = \rho g \Delta H$$

If this equation is used to equate Equations 1 and 2, the density must be 62.0933 lbm/ft³ to achieve equality. Hence any other density used in AFT Fathom will result in a pressure drop which does not equal to that of the Equation 1 NFPA formulation. Using the ASME Steam Tables, a density of 62.0933 lbm/ft³ at 1 standard atmosphere occurs at a temperature of 91.78 F.

Matching AFT Fathom Hazen Williams to NFPA Hazen-Williams

Users who want their results to match the NFPA version in Equation 1 can do so by one of the following two options:

- Use a density of 62.0933 lbm/ft³ in all calculations, or
- Use the desired density and then apply a [design factor](#) on each pipe to correct the pressure drop for density. The design factor should be equal to 62.0933 lbm/ft³ divided by the density on the [Fluid panel](#) in [Analysis Setup](#). If variable properties are used, the design factor must be varied on a pipe-by-pipe basis.

Related Topics

NFPA Panel.....	175
Equivalent Length.....	267

Heat Transfer Sectioning Panel

If [heat transfer](#) is enabled, the number of thermal sections that are used to calculate heat transfer in Pipes and Heat Exchangers can be modified in the Heat Transfer Sectioning panel, shown in Figure 1.

Because the amount of heat transfer is dependent on the temperature difference at any given point, an overall average would not give an accurate estimate. Sectioning heat transfer components increases the accuracy of the calculation due to the changing thermal profile. Very large thermal changes may require additional sections.

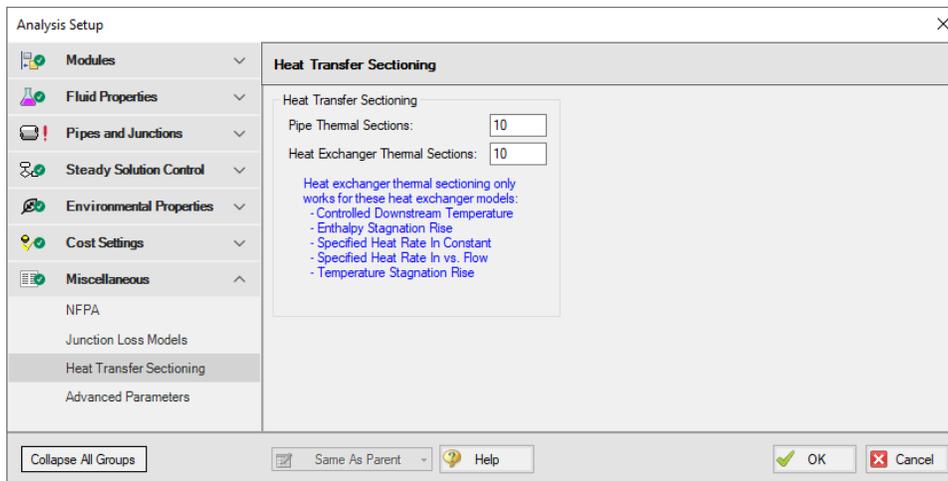


Figure 1: Default state of the Heat Transfer Sectioning panel

Related Topics

Miscellaneous Group.....	175
Advanced Parameters Panel.....	179
NFPA Panel.....	175

Advanced Parameters Panel

The Advanced Parameters panel, shown in Figure 1, of [Analysis Setup](#) offers various settings intended only for troubleshooting purposes.

These parameters should not be changed under normal circumstances and will significantly impact results.

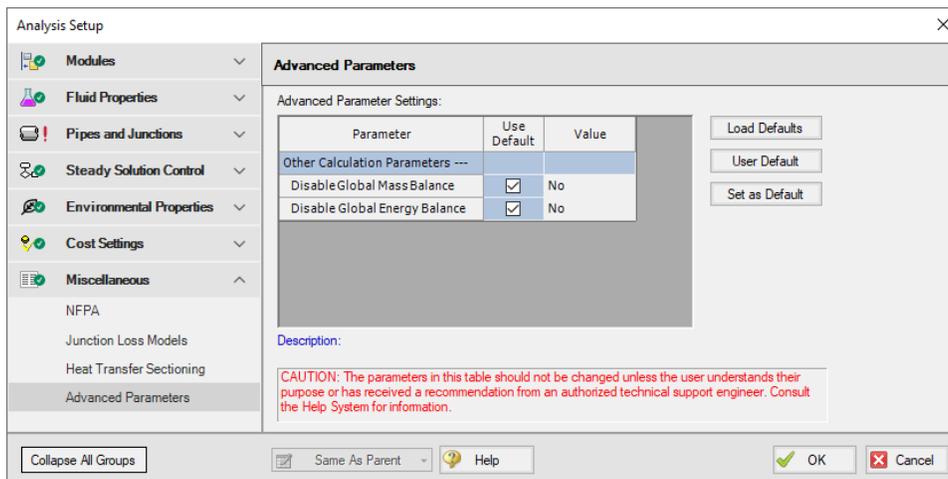


Figure 1: Default state of the Advanced Parameters panel

Related Topics

Miscellaneous Group.....	175
Heat Transfer Sectioning Panel.....	178
NFPA Panel.....	175
Global energy balance disabled by user in Advanced Parameters Panel.....	774
Global mass balance disabled by user in Advanced Parameters Panel.....	774

Working with Pipe and Junction Objects

This section focuses on interactions with the Pipe and Junction objects in the AFT Fathom workspace. Topics include how to create pipe and junction objects, how to alter junctions already on the Workspace, and how to ensure the inputs for objects are both correct and complete.

Contents

Selecting and Finding Objects	181
Pipe Drawing Modes	186

Selecting and Finding Objects

AFT Fathom has a number of methods to assist in selecting multiple objects at once as well as finding objects for recall later. The contents of this section can be broken down into the two categories shown below. Follow the links to learn more.

- Methods for [selecting objects](#)
 - Select Flow Path
 - Select Special
 - [Groups](#)
- Methods for [finding objects](#)
 - Workspace Find
 - Workspace Search
 - Find from Model Data or Output
 - Jump Feature
 - Bookmarks

Selecting Objects

A single left click on any object will select that object. Selecting multiple objects can be accomplished by:

- Holding the shift or control key and left clicking on objects
- Dragging the cursor over the objects, creating a selection area
 - Dragging left-to-right creates a selection area that will only select objects entirely bounded by the area
 - Dragging right-to-left creates a selection area that selects any objects completely or partially inside the area
- Using Select All from the Edit Menu, right-clicking on Workspace to open context menu, or keyboard shortcut (Ctrl-A)

- Using Select Last Selection from the Edit Menu or with F12 to reselect the previously selected objects
- Using **Select Flow Path**
- Using **Select Special**
- By selecting a [Group](#)

Select Flow Path

A flow path is a sequence of pipes and junctions where there is no branching or flow sources or sinks.

The pipes and/or junctions in a flow path may all be easily selected using the Select Flow Path item on the [Edit menu](#), [Toolbar](#), or the right-click context menu.

If a junction is selected which has multiple flow paths attached to it (e.g., a branch with 3 or more pipes) then objects in each of the connected flow paths will be selected. This feature is very useful in duplicating or deleting sequences of pipes and junctions. It can also be used to quickly define groups for graphing or for selected Output or Model Data listings.

Select Special

Select Special is a tool for selecting or deselecting [objects](#) based on certain criteria. It can be opened from all windows/menus that give you the option to choose what pipes and/or junctions are selected. These windows/menus include the Output Control window, the Model Data window, the Layer Settings window, the Select Graph Data window, the [Edit menu](#), right clicking on the [Workspace](#), or pressing F6 on the keyboard.

Once Select Special is open, all pipe names and numbers and junction names and numbers are shown in numerical order under Pipe Selections and Junction Selections, respectively. At the top of the Select Special window is the Selection Type option. This option allows you to choose what criteria you would like to use to select certain pipes and junctions. You can choose from the following criteria to select certain pipes and/or junctions.

- Selected on Workspace
- Closed in Model
- Special Conditions Set
- Number Range
- Junction Type
- Pipe Input Properties
- Zero-Length Connectors
- Notes
- Undefined Objects
- User-Colored Pipes
- Transient Data Exists (XTS only)
- Groups

- Used as Goal or Variable
- Variable Fluid Properties
- Pipe Output Properties (Only available when output is available)
- Junction Output Properties (Only available when output is available)

Choosing certain Selection Types will prompt a second or sometimes third option below the Selection Type to allow you to further specify the type of pipe or junction you want. For example, selecting Junction Type in the Selection Type box will prompt the Junction Type box to appear. The Junction Type box will allow you to specify what type of junctions you would like to select.

By default, the Selection Type is Selected on Workspace and any pipes or junctions selected on the Workspace are selected in the Select Special window with a black check mark next to the box corresponding to the pipe or junction.

Once you have fully specified the Selection Type, you must click Select Pipes and/or Select Junctions to automatically select the corresponding objects.

The selected pipes and junctions under Pipe Selections and Junction Selections do not automatically become unselected when you switch from one Selection Type to another. Therefore, you must first choose Deselect Pipes and/or Deselect Junctions located above the Pipe Selections and Junction Selections boxes, respectively. For example, if you select all of the pipes on the Workspace prior to opening the Select Special window, all of the pipes will be selected in the Select Special window when you open it. If you then choose Junction Type as the Selection Type and then specify All Junctions as the Junction Type, all junctions will be selected under Junction Selections, but all pipes previously selected will also remain selected. In this example, if you would like only junctions to be selected with none of the selected pipes on the Workspace to be selected in the Select Special window, first click the Deselect Pipes button, which is located above the Pipe Selections list.

Options to select all pipes/junctions, to deselect all pipes/junctions, and to invert the selected pipes/junctions are located below both Pipe Selections and Junction Selections boxes. Inverting the pipes/junctions deselects all currently selected pipes/junctions and selects all currently deselected pipes/junctions. This feature is helpful if you want to select all pipes/junctions that **do not** fit a certain criteria. For example, if you want to select all open objects in the model, you could specify Closed in Model in the Selection Type box, then click Select Pipes and Select Junctions. All pipes and junctions that are closed in the model will be selected. Because you want to select all open pipes and junctions in the model but only the closed pipes and junctions are selected, you would simply click Invert below both the Pipe Selections and Junction Selections lists to deselect all closed pipes and junctions and select all open pipes and junctions.

Related Topics

Selecting and Finding Objects	181
Finding Objects	184
Groups	185

Finding Objects

Workspace Find

The Find window utility helps you find a particular pipe or junction by its ID number and name. The Find utility becomes more useful for large models.

You can access the Find utility by selecting Find on the [View menu](#), by clicking the binocular button on the [Toolbar](#), or by pressing the CTRL + F keys on the keyboard. After you double click on the pipe or junction you are searching for in the Find utility or when you click the Find button, the Workspace is moved so that the pipe or junction is located near the center of the screen. Additionally, the pipe or junction you are searching for is selected in the Selected Items color specified in the [User Options](#) window in the [Tools menu](#). By default, this color is red.

Workspace Search

The Search window utility helps you search for a particular pipe or junction by its ID number, name, or notes you have inputted in the properties window. The Search utility becomes more useful for models that expand over multiple Workspace pages.

You can access the Search on the Common [Toolbar](#). The search feature will list all objects related to your search term beneath itself in a list of search results. You can hover over each search results to see a preview of input and output (if the model has been run). Clicking on or pressing enter while the object is highlighted in the search results will select the object in the workspace and center the focus of the workspace on the object.

Find From Model Data or Output

Objects can be quickly located on the Workspace by right-clicking on a cell in Model Data or row header in Output and selecting "Find on Workspace..." the view will change to show the Workspace, with the object of interest selected.

Jump Feature

By clicking the Jump button in a Properties window, you can move directly to the Properties window of another a pipe or junction that you specify. The Jump button is located in the upper-right corner of any Properties window.

Bookmarks

Use the Bookmark tools on the [View menu](#) to set Bookmarks and to display different bookmarked pipes or junctions.

Bookmarks can be added on the [Workspace](#) in relation to individual pipes or junctions. You can quickly navigate around the Workspace by moving from Bookmark to Bookmark.

Related Topics

Selecting and Finding Objects	181
Selecting Objects	181
Groups	185

Groups

Groups of [pipes and junctions](#) can be named for later recall. This is done using the Group tools on the Edit menu. The easiest way to create a named Group is as follows:

1. First select the pipes and junctions you want in the group.
2. Choose Groups -> Create from the Edit menu or Toolbar.
3. You will be prompted for a name for the group. Enter a meaningful name and click OK.
4. The Group Manager then opens. The selection of pipes and junctions in the Group will match those currently selected on the Workspace. If you want to modify the selections further, you can do it here.

You can include Subgroups inside of your Groups. Subgroups are existing Groups. If a Group contains one or more Subgroups and the Group is selected on the Workspace, then all pipes and junctions in the Group and Subgroup(s) will be selected. Additionally, if you include a Subgroup in your Group and then later change the contents of the Subgroup, the Group that contains the Subgroup will also be changed.

After you have created a Group, you can then quickly reselect that Group of pipes or junctions on the Workspace by choosing Group -> Select from the Edit menu or Toolbar and choosing the name of the Group.

The Group Manager can be opened by choosing Group -> Manager from the Edit menu or Toolbar. The Group Manager allows you to modify, create, delete or rename Groups.

Another place that Groups can be selected from is in the [Select Special](#) window.

Related Topics

Selecting and Finding Objects	181
Selecting Objects	181
Finding Objects	184

Related Examples

NFPA Firewater System	934
Spray Discharge System - GSC	1019
Pump Sizing and Selection with Flow Control Valves - GSC	1001
NFPA Firewater System - GSC	1032

Pipe Drawing Modes

AFT Fathom has a selection of pipe drawing modes. To select or change a pipe drawing mode, navigate to Arrange -> Pipe Drawing Mode. These pipe drawing modes are:

- [2D Freeform](#)
- [2D Orthogonal](#)
- [Isometric](#)

A second function called **Snap to Grid** is highly integrated with Pipe Drawing Modes and will alter how each pipe drawing mode works. To select or change a Snap to Grid option, navigate to Arrange -> Snap to Grid. The Snap to Grid options for the 2D modes are on and off. Snap to Grid options for Isometric are more extensive. These snap to grid options are:

- Snap to Isometric Grid Lines
- Snap to Grid Points
- Do Not Snap

2D Freeform

This Pipe Drawing Mode allows placing pipes or junctions anywhere on the Workspace, with pipes at any angle. Snap to Grid can be turned on with this mode to force the junctions to be at grid intersection points.

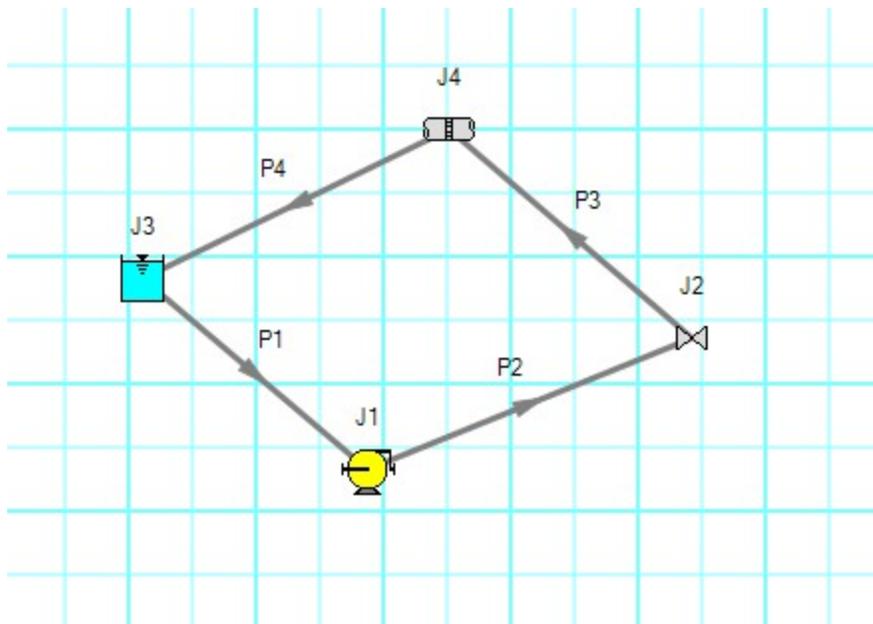


Figure 1: 2D Freeform with Snap to Grid off

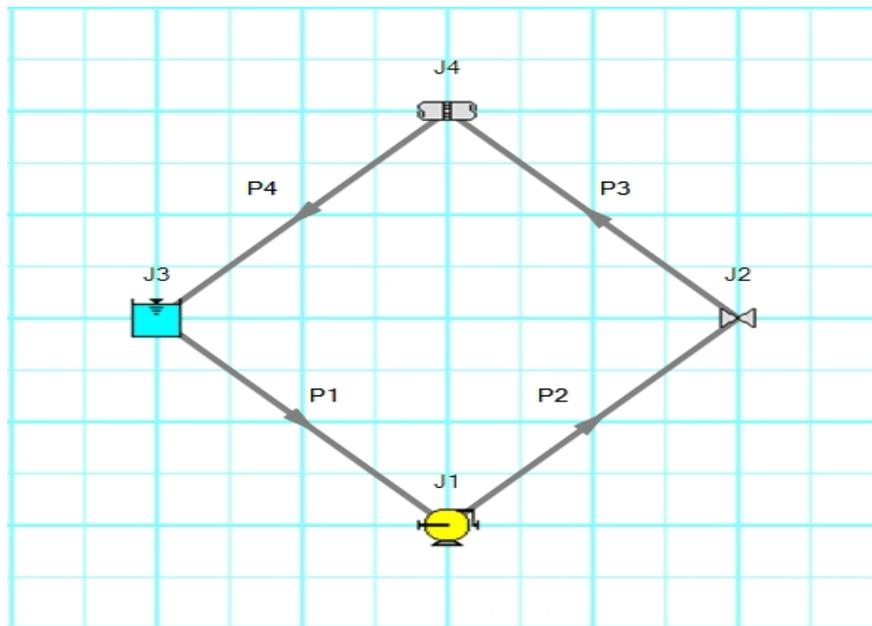


Figure 2: 2D Freeform with Snap to Grid on

2D Orthogonal

The 2D Orthogonal Pipe Drawing Mode forces all pipes drawn in the model to be either horizontal or vertical. Segments are automatically inserted into pipes to enforce drawing only along the horizontal and vertical directions.

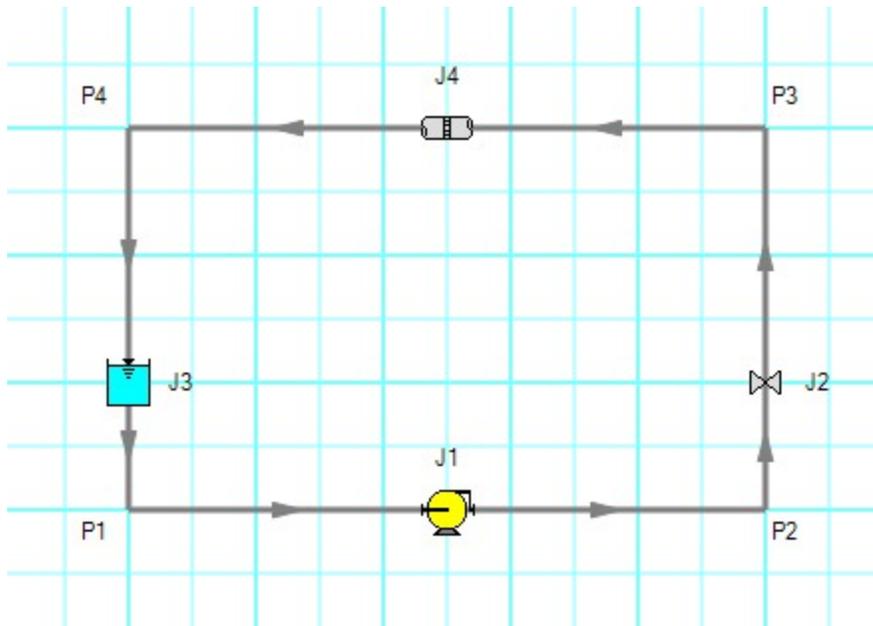


Figure 3: Model in 2D Orthogonal mode with Snap to Grid on

Orthogonal drawing can create some confusion for the user because segments from different pipes may overlap each other and hence become indistinguishable.

- Use the Add/Remove Segments button to add new segments
- Use a [fine grid](#) and the Snap to Grid features
- If you have a junction with several connecting pipes that interfere, enlarge the junction to 150% or 200% of normal size using the [Workspace Layers](#)

Isometric

When modeling, it may be convenient or desirable to demonstrate the three dimensional nature of a system. A typical 2D grid has two sets of grid lines representing the x and y axes. An isometric grid has three sets of grid lines that are offset by 60°, representing the x, y, and z axes. It is ideal to set the grid type desired before building a model to avoid the pipe and junction locations from being rearranged in undesired ways. It is also very helpful to show the Grid (Arrange -> Show Grid) when building an isometric model.

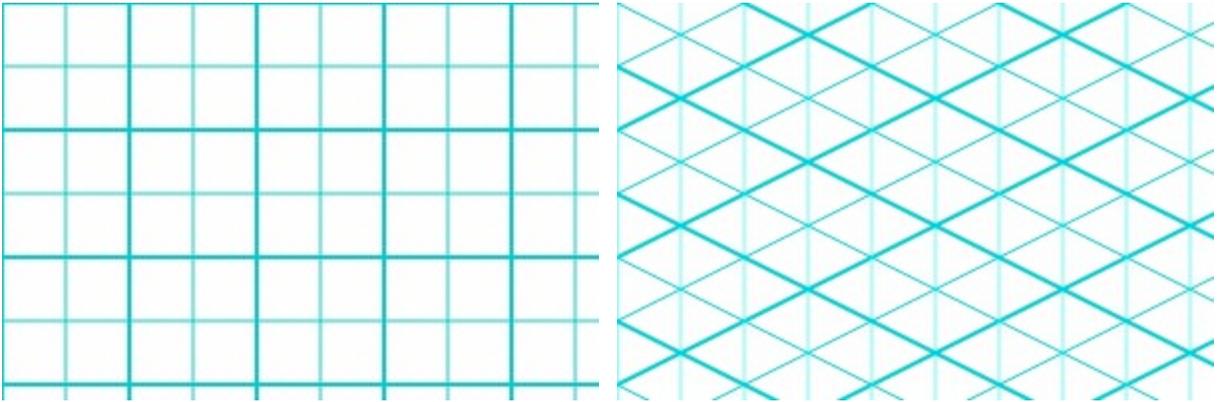


Figure 4: 2D (left) and isometric (right) grids

Drawing Pipes in Isometric Mode

How pipe drawing in isometric mode functions will change depending on the selected snap to grid option. When Snap to Isometric Grid Lines is selected, the pipes are automatically segmented to lie on the grid lines as shown below. Dragging one of the handles on the pipe shows a preview line of where the pipe will be placed if adjustment needs to be made. The placement of the preview line can be cycled through all possibilities by pressing the arrow keys or by scrolling the mouse scroll wheel. Holding the alt key while dragging a pipe endpoint will automatically add a new segment to the pipe - the placement of the line can be controlled with the arrow keys or scroll wheel as before.

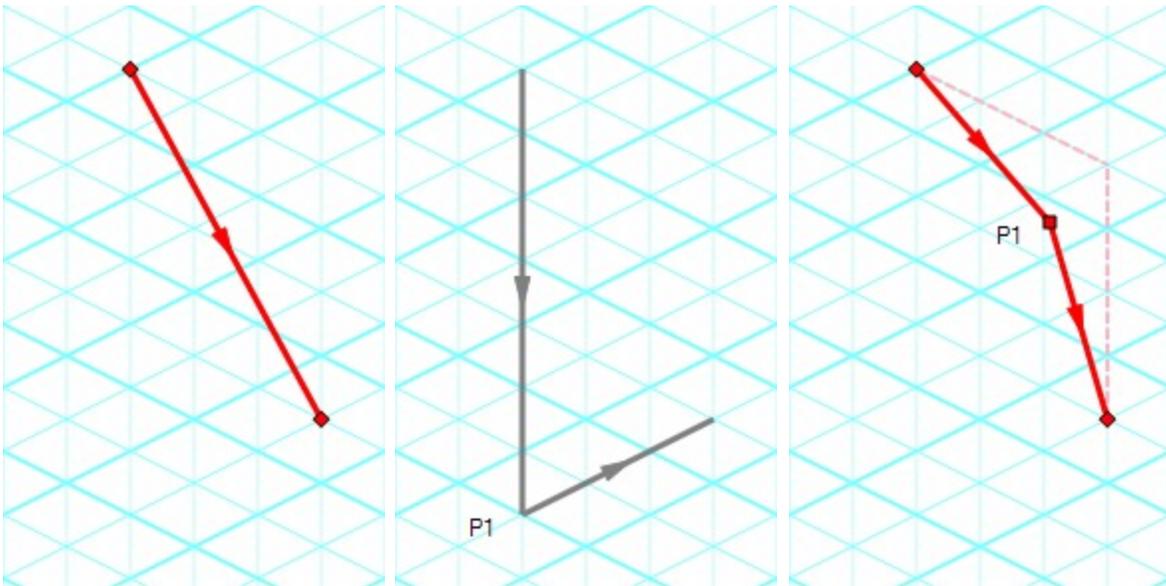


Figure 5: Automatic layout of isometric pipes

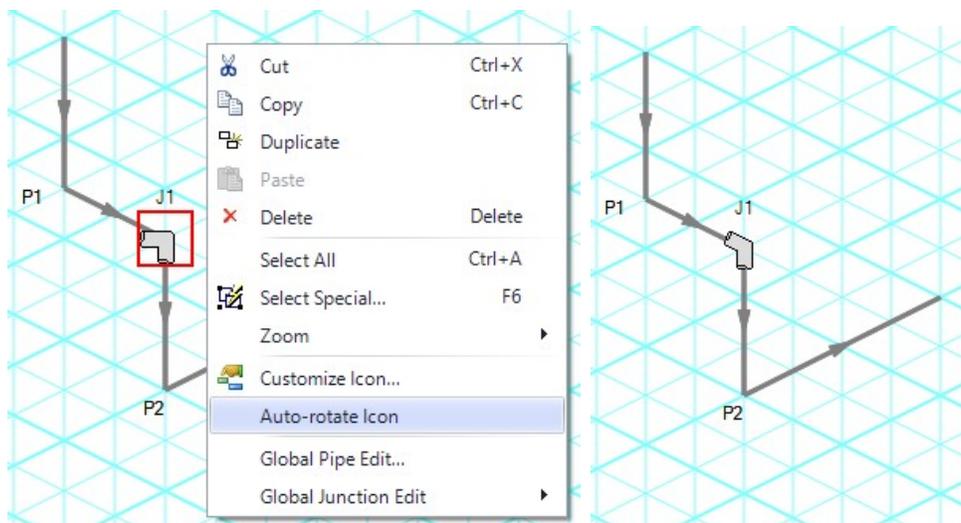
The Snap to Grid Points option will force the end points of the pipes to snap to the points where the grid lines intersect without adding segments to it. This is akin to the 2D Freeform mode while Snap to Grid is on.

The Do Not Snap option will allow the end points of the pipes to be placed anywhere within the Workspace.

Placing Junctions

Placing junctions onto the Workspace follows the usual rules; however, the visual appearance of the icon is more complex than in an orthogonal grid. When icons are added after pipes are placed, AFT Fathom will rotate the icon automatically following the Workspace arrangement. When pipes are connected to icons already placed on the Workspace, right-click and select Auto-rotate Icon to adjust the icon. For manual control of icon appearance, right-click the junction and select Customize Icon to choose the desired appearance.

Note: The Rotate button in the Customize Icon window can be used to find the appropriate icon orientation.



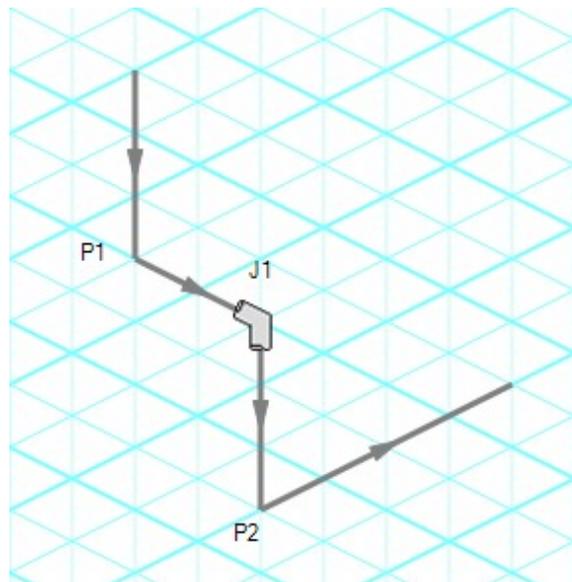
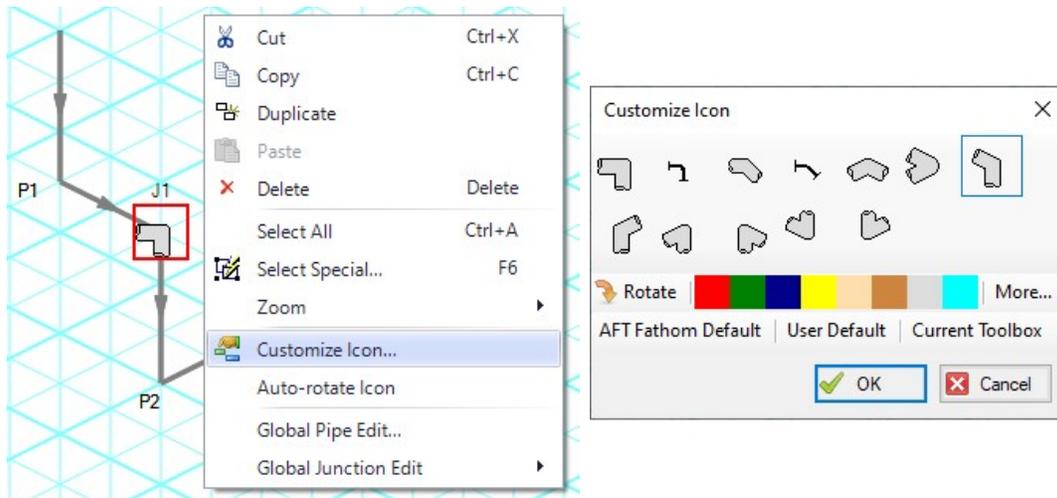


Figure 6: Icon rotations in isometric space

Related Topics

Working with Pipe and Junction Objects.....	181
Selecting and Finding Objects.....	181
Global Edit.....	193

Related Examples

Spray Discharge System.....	874
-----------------------------	-----

NFPA Firewater System 934

Global Edit

AFT Fathom has the powerful capability to apply input parameters across a wide range of objects. This can be done for pipes by using the [Global Pipe Edit](#) or across junctions by using the [Global Junction Edit](#).

Global Pipe Editing

The Global Pipe Edit window allows you to edit data for multiple pipes at the same time. A Guide is available at the top of the Global Edit window to walk you through the global edit process.

Here is how you use it:

1. Open the Global Pipe Edit window from the Edit menu, from the Workspace toolbar, or from the right-click menu on the Workspace.
2. Select the pipes you want to edit from the Pipe list on the left (if pipes were selected in the Workspace, they will be automatically selected here).
3. Click the Select Pipe Data button. This will open a template [Pipe Properties Window](#) in which you can enter the data you want to change on each of the selected pipes. Alternatively, if you want to copy the data from an existing pipe, you can choose "Existing Pipe", then select the desired pipe number and skip to step 5.
4. Enter the pipe data you want to change and click the OK button in the Pipe Properties window.
5. The Global Pipe Edit window now lists the parameters to change in a list on the right. Select the parameters you want to change.
6. Click the Apply Selections button.

Note: You cannot change some data, like the pipe ID number, using this global method.

Note: There is no undo for global pipe editing. Therefore, you cannot revert your changes once you have exited the Global Edit window. However, hitting Cancel on the Global Edit window will undo any changes that were just applied.

Related Topics

Global Edit	193
Global Junction Editing	193

Global Junction Editing

The Global Junction Edit window allows you to edit data for multiple junctions at the same time. A Guide is available at the top of the Global Edit window to walk you through the global edit process, or the instructions are provided below in this topic.

Junction data falls into the following two broad categories:

- Common Data- data that applies to all junctions regardless of junction type; examples include elevation and initial pressure data.

- Junction Specific Data- data that applies only to certain junctions; examples include pump curves and number of heat exchanger tubes.

To open the Global Junction Edit window, browse to the Global Junction Edit option on the Edit menu or the right-click menu. From there, choose the category of junction you want to edit. The available categories include those junctions selected on the Workspace, any junction type used in the model, and common junction data. Global Junction Edit can also be launched by right-clicking on the Workspace.

Note: There is no undo for global junction editing. Therefore, you cannot revert your changes once you have exited the Global Edit window. However, hitting Cancel on the Global Edit window will undo any changes that were just applied.

Editing Common Junction Data

You can open the Global Junction Edit window for common data editing in two ways. First, you can choose Common Data from the Global Junction Edit menu selection on the Edit menu, or you can choose one of the other specific junction types on the menu and then choose the Common option for the Junction Template in the upper left.

Here is how to globally edit junction common data:

1. Open the Global Junction Edit window.
2. Select the Common option for Junction Template in the upper left.
3. Select the junctions you want to edit from the Junction list on the left. Note that all junctions in the model are shown, no matter what the junction type.
4. Click the Select Common Junction Data button. This will open the Common Junction Data Edit window in which you can enter the data you want to change on each of the selected junctions.
5. Enter the junction data you want to change and click the OK button in the Junction Properties Window.
6. The Global Junction Edit window now lists the parameters to change in a list on the right. Select the parameters you want to change.
7. Click the Apply Selections button.

Note: You cannot change some data, like the junction ID number, using this global method.

Editing Specific Junction Data

You can open the Global Junction Edit window for specific data editing in two ways. First, you can choose the junction type from the Global Junction Edit menu selection on the Edit menu, or you can choose Common Data on the menu and then choose the desired junction template, or Existing Junction data for the desired junction type.

Here is how to globally edit specific types of junctions:

1. Open the Global Junction Edit window.
2. Select the Junction Template option in the upper left. Alternatively, if you want to copy the data from an existing junction, you can choose "Existing Junction", then select the desired junction number and skip to step 6.
3. Select the type of junction you want to edit in the dropdown list in the upper left. For example, you could select the Spray Discharge junction.
4. Click the Select Spray Discharge Data button (the button name will change based on the type of junction selected). This will open the appropriate junction Properties Window, in this case the Spray Discharge Properties Window.
5. Enter the junction data you want to change and click the OK button in the Spray Discharge Properties Window (or whatever window type is displayed).
6. The Global Junction Edit window now lists the parameters to change in a list on the right. Select the parameters you want to change.
7. Select the junctions you want to edit from the Junction list on the left. Note that only junctions of type selected in the upper left are shown (Spray Discharge junctions in this case).
8. Click the Apply Selections button.

Related Topics

Global Edit	193
Global Pipe Editing	193

Toolbox

The Toolbox is located on the left of the [Workspace](#) window by default, but can be undocked by either dragging or right-clicking the heading and selecting Float. The assortment of tools allows you to create pipes, junctions, and annotations.

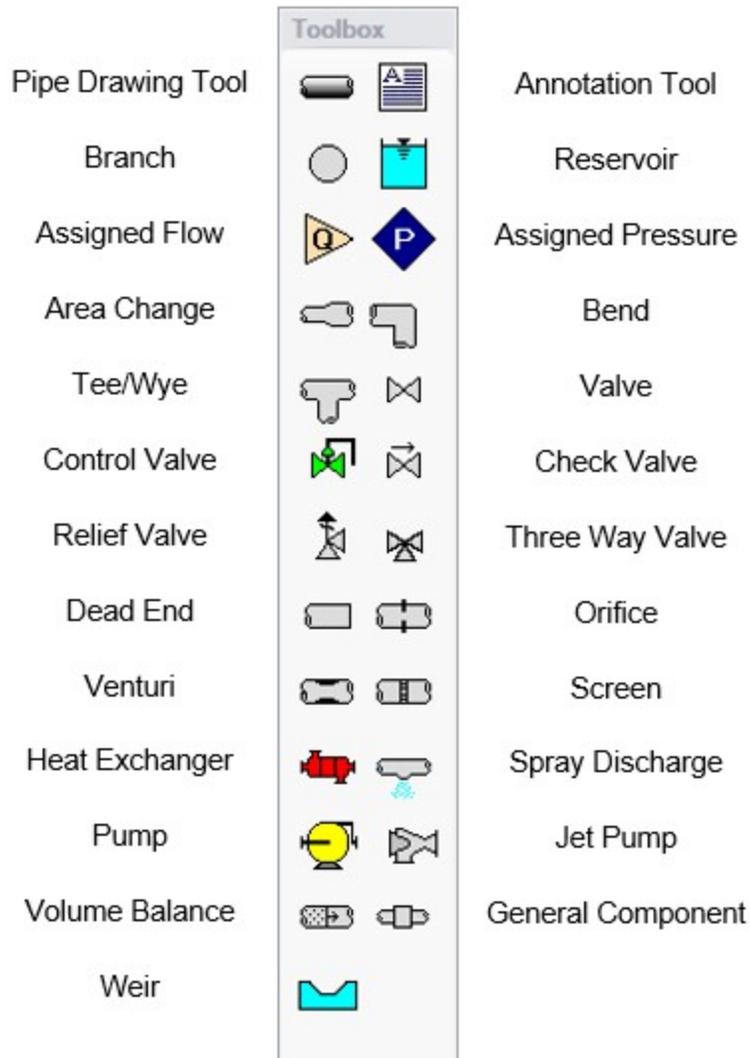


Figure 1: Toolbox items

Custom Junction Icons

Right-clicking a junction icon in the Toolbox will open the Customize Icon window. A different icon can be selected from the predefined assortment, or a custom image can be uploaded using the Add Image button. This functionality enables you to use standard P&ID icons or custom images to visually represent your equipment. By using recognizable icons, you can improve the clarity and comprehension of model results, especially for those who may not be familiar with the default AFT junction icons.

Note: The [Annotation Tool](#) also supports triangle and cloud shapes which can be used in combination with the Notes section on the Scenario tab of the [Quick Access Panel](#) to designate revisions of the model file, similar to a typical P&ID drawing.

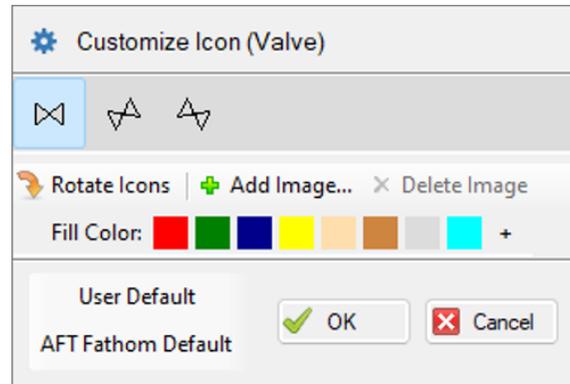


Figure 2: The *Customize Icon* window allows a custom image to be applied to any junction

Related Topics

Creating Pipes - the Pipe Drawing Tool.....	197
Creating Junctions.....	202
Keyboard Shortcuts for Workspace.....	404

Creating Pipes - the Pipe Drawing Tool

The tool in the upper left corner of the [Toolbox](#) is the Pipe Drawing tool. With this tool you can draw new pipes on the Workspace, starting and ending at any desired location.

To draw a pipe, click the Pipe Drawing tool. The mouse pointer will change to a crosshair when you move it over the Workspace. Position the cursor at the pipe's starting point, hold down the left mouse button while dragging the cursor to the pipe end point, then release the mouse button.

To draw multiple pipes, double click the Pipe Drawing tool. This will keep the tool engaged until you click the Pipe Drawing tool again or press the escape key.

The length of the line on the Workspace has no relationship to the physical length of the pipe. The Workspace pipe is merely an abstract representation of a pipe, however, each pipe drawn must have a visible length on the Workspace. If the pipe drawn does not have a visible length, AFT Fathom eliminates the line immediately after you release the mouse.

When a single pipe is selected, two black squares (or more if the pipe is segmented), called *handles*, appear. When the cursor is placed over a pipe handle it changes to a crosshair and the pipe handle can be moved or stretched by dragging with the left mouse button.

Another way to create new pipes is to duplicate existing pipes with the Duplicate or the Copy and Paste features found on the [Edit menu](#). These features can also be accomplished both from the right-click menu on a selected pipe as well as the Workspace toolbar. When a new pipe is derived from a previous pipe, the new pipe retains the input associated with the original pipe.

Each pipe must be connected to two junctions. There are no exceptions to this rule.

Pipe ID Numbers

When a new pipe is created, it is assigned a default ID number. When the pipe is displayed on the Workspace, the ID number is preceded by the letter "P" to indicate that it is for a pipe. From the [Pipe Properties Window](#), you can change the ID number to any desired number greater than zero and up to 99,999, but the number must be unique to the pipe and cannot be replicated in any other pipes on the Workspace. The pipe number remains with the pipe in descendant scenarios in the [Scenario Manager](#). An arrow near the ID number indicates the [reference flow direction](#) for the pipe. The reference positive flow direction can be reversed by choosing [Reverse Direction](#) from the Arrange menu or by selecting the reverse direction button on the [Toolbar](#).

Tip: The pipe or junction ID number text can be dragged to a new location to improve visibility. The new location is saved with the model. If you want the text of one or more pipes and/or junctions returned to the default location, select those pipes/junctions on the Workspace and choose Reset Label Locations from the [Arrange menu](#). This will open the Reset Label Locations window. From the Reset Label Locations window, click the Reset Labels For Currently Selected Pipes and Junctions. If you want all pipe and junction text returned to the default locations, choose Reset All Labels from the Reset Label Locations window.

Pipe Segments

A pipe can be segmented or "bent" into numerous sections. This only affects the visual representation and does not affect the calculated results. It does not add an "elbow" to the model or the resulting pressure drop effects of an elbow. This feature is useful in graphically bending the pipe around another part of the model or graphically showing that there are embedded elbows as [fittings & losses](#).

You can add a segment to a pipe in several ways. The easiest way to add a pipe segment is to first select the pipe, then click on the +/- image on the Add/Remove Segment icon from the Toolbar. A segment will automatically be added. One or more segments can be added by clicking on the down-facing arrow on the Add/Remove Segment icon from the Toolbar, and then choosing the number of segments to be added. Alternatively, you can first select the pipe, then select Pipe Segments in the Arrange menu. A dropdown list will appear that allows you to choose Add a Segment or Add Multiple Segments.

The easiest way to remove a segment is to first select the pipe, then click on the down-facing arrow on the Add/Remove Segment icon from the Toolbar. A drop down list will appear that will allow you to choose Remove Segment. If you wish to remove all segments, click Remove All Segments. Alternatively, you can first select the pipe from which you want to delete one or more segments, then select Pipe Segments in the Arrange menu. A dropdown list will appear that allows you to remove one or all segments by clicking on either Remove Segment if you want to remove one segment, or by clicking on Remove All Segments if you want to delete all segments.

Related Topics

Toolbox.....	195
--------------	-----

Creating Junctions.....	202
Keyboard Shortcuts for Workspace.....	404
Pipes.....	214
Joining or Splitting Pipes.....	199
Connecting Objects.....	203
Moving Objects.....	204
Editing Objects.....	205
Defining Objects.....	206
Highlight Required Input.....	208
Inspecting Objects.....	209
Notes.....	210
Status.....	211

Related Examples

Beginner - Three Reservoir	823
----------------------------------	-----

Joining or Splitting Pipes

When pipes are present in the [Workspace](#), users can both combine and split up pipes. This can be done either to clean up the model visually or to further manipulate the model.

Joining Pipes

When multiple pipes are placed in the workspace separated by junctions without having other [flowpaths](#) splitting off, those pipes and intermediate junctions can be combined into a single pipe. The new pipe will have a total length equal to the sum of the combined pipe lengths, and any losses associated with the intermediate junctions will be summed and entered as additional [Fittings & Losses](#). It will also have the elevation profile of the original pipes and junctions entered as [Intermediate Pipe Elevations](#).

The pipes being combined must have the same material, size, type, and friction model to be combined. In addition, the pipes must be pointing in the same direction.

Joining pipes may cause some junction data to be lost such as special conditions, loss factors for reverse flow, and ignored transients ([XTS module](#)).

To join pipes and junctions in the model, select the "Join Pipes/Junctions" option from the Edit menu. The Join Pipes window will be shown with more information on pipes that can be combined in the model. The behavior of the Join Pipes/Junctions window will vary depending on if junctions are selected on the Workspace when the Join Pipes/Junctions option is selected.

Note: For maintenance releases prior to September 2022 the Join Pipes/Junctions window did not exist. Instead pipes and junctions selected in the Workspace would be joined automatically when the "Join Selected Pipes/Junctions" option was selected in the Edit menu.

- If junctions are selected on the Workspace, the Join Pipes/Junctions window will open with only the selected junctions shown in the list.

- If no junctions are selected on the Workspace, the Join Pipes/Junctions window will open with all junctions that can be combined shown in the list. Any junctions that can be combined and won't cause a loss in data at that junction will be selected automatically.

Once the Join Pipe/Junctions window is opened the selections can be modified before joining the pipes. The options below the table can be enabled/disabled in order to filter the shown junctions based on if they are selected on the workspace, if they will lose data when combined, or if they can't be combined.

If any junctions are selected that would lose data when combined, an additional button will appear near the bottom of the window that allows junctions that would have data loss to be deselected. It is recommended to select this button before joining the pipes/junctions unless you are certain the lost data is no longer required.

To join the pipes at the junctions selected in the Join Pipes/Junctions window, click the Join Selected Junction(s) button at the bottom of the window.

Splitting Pipes

An existing pipe in the workspace can be split into multiple pipes two different ways. First, [holding the Shift](#) key while placing a junction along the pipe will split the pipe in two and insert the selected junction at the specified length. Any Fittings & Losses defined in the pipe can be placed in either of the split pipe segments.

Second, pipes with Intermediate Pipe Elevations can be split into separate pipes at any or all of those intermediate elevations. This can be done by selecting a pipe with Intermediate Pipe Elevations and clicking Edit -> Split Pipe at Intermediate Elevations. A window will open to select which intermediate elevations to make into new junctions.

The pipe can be split at all intermediate elevations, or at selected locations. This is specified by checking the box next to each elevation point the pipe should be split at. When the pipe is split, a branch will be created for each of the selected elevation points. If desired, a name can be entered for each new branch in the New Junction Name column of the table. Any points not selected will be retained in the intermediate elevation table of the appropriate pipe in the Workspace.

Note: When splitting a pipe at intermediate elevations, the pipe in the Workspace must be long enough to place the necessary number of branches in the pipe before the split can be completed. If insufficient space is available, the user will be required to exit the Split Pipe at Intermediate Elevations window and rearrange the model to create space before Fathom can perform the split action.

Split Pipe At Intermediate Elevations

Select the intermediate elevations where you would like to split the pipe and add a Branch junction.
(NOTE: List does NOT include endpoints)

Split At	Total Length (feet)	Segment Length (feet)	Elevation (feet)	New Junction Name
<input type="checkbox"/>	100	100	-9	
<input type="checkbox"/>	200	100	-15	
<input type="checkbox"/>	300	100	-12	
<input type="checkbox"/>	400	100	-3	
<input type="checkbox"/>	500	100	0	
<input type="checkbox"/>	600	100	5	
<input type="checkbox"/>	700	100	12	
<input type="checkbox"/>	800	100	16	
<input type="checkbox"/>	900	100	20	
<input type="checkbox"/>	1000	100	15	
<input type="checkbox"/>	1100	100	12	
<input type="checkbox"/>	1200	100	5	

All Invert None 0 of 12 selected. Edit Table

OK Cancel Help

Figure 1: Split Pipe at Intermediate Elevations window

For pipes with a large number of intermediate elevations, it may be more convenient to define the split points in a spreadsheet. This can be done from the Edit Table button at the bottom of the window by selecting Export to File. This will export the table to a *.csv table which can be opened in Excel, as is shown in Figure 2.

The table can be updated by changing the "Split At" column value to Yes for each location that the pipe will be split at. Branch names can also be entered under "New Junction Name" column for the appropriate points. Note that it is important to not change any of the Length or Elevation columns in the *.csv file, as this will prevent the file from being imported. To import the file into Fathom, select "Import From File" from the Edit Table button, then browse to the updated *.csv file.

	A	B	C	D	E	F	G
1	Split At	Total Length (feet)	Segment Length (feet)	Elevation (feet)	New Junction Name		
2	No	100	100	-9			
3	No	200	100	-15			
4	No	300	100	-12			
5	No	400	100	-3			
6	No	500	100	0			
7	No	600	100	5			
8	No	700	100	12			
9	No	800	100	16			
10	No	900	100	20			
11	No	1000	100	15			
12	No	1100	100	12			
13	No	1200	100	5			
14							

Figure 2: Exported *.csv file for splitting a pipe at intermediate elevations

Related Topics

Pipes.....	214
Creating Pipes - the Pipe Drawing Tool.....	197
Connecting Objects.....	203
Moving Objects.....	204
Editing Objects.....	205
Defining Objects.....	206
Highlight Required Input.....	208
Inspecting Objects.....	209
Notes.....	210
Status.....	211

Creating Junctions

Below the pipe drawing and the annotation tools on the [Toolbox](#) are junctions, which allow you to model a large variety of pipe system components. To add a junction to a model, drag the desired icon from the Toolbox and drop it anywhere on the [Workspace](#).

To add multiple of the same junctions to a model, double click the desired icon and click the workspace to add the icon to the desired location. The selected junction icon will remain engaged until you click the junction icon again or press the escape key.

As you move the mouse pointer over the Toolbox, a tooltip identifies the type of junction under the mouse pointer.

When you duplicate a junction using the Duplicate, Copy, or Paste features on the [Edit menu](#), the new junction retains the input associated with the original junction.

Junction ID Numbers

Junctions are shown on the Workspace with default ID numbers, which you can change in the [Junction Properties Window](#). The letter "J" precedes the ID number to signify that it is for a junction. From the junction's Properties window, you can change the ID number to any desired number greater than zero and up to 99,999, but the number must be unique to the junction and cannot be replicated in any other junctions on the Workspace. The junction number remains with the junction in descendant scenarios as long as the junction type is not changed.

Identical ID numbers can be assigned to both a pipe and a junction because the "P" or "J" will distinguish the two.

Unlike pipes, junctions do not have reference positive flow directions. Junctions for which the flow direction is important (such as control valves) derive the flow direction from the flow direction of the connecting pipes.

Related Topics

Toolbox.....	195
Creating Pipes - the Pipe Drawing Tool.....	197
Keyboard Shortcuts for Workspace.....	404

Related Examples

Beginner - Three Reservoir	823
----------------------------------	-----

Connecting Objects

During construction of a new model, objects can be placed anywhere on the [Workspace](#). In order to assemble a model that is ready to submit to the Solver (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system), you must connect the objects properly.

Remember that **connectivity only exists between [junctions and pipes](#)**. There are no junctions that connect to junctions, and no pipes that connect to pipes.

However it is possible to represent two junctions that are directly connected, such as two components that are welded together by using a pipe with the [Model As Zero-Length Connector](#) option.

AFT Fathom will attempt to automatically connect pipes and junctions when they are moved. This eliminates unnecessary opening of the pipe and junction Properties windows. If Fathom cannot make the connection automatically, then the following three steps may be required to establish a connection between a junction and a pipe:

1. Graphically connect the objects on the Workspace. The pipe endpoint must terminate within the boundaries of a junction icon.
2. After the pipe is graphically connected to two junctions, double-click the pipe to open its [Properties Window](#), then click OK to accept the connected junctions as determined by AFT Fathom. It is most

efficient to do this when you are entering the pipe's data in its Properties Window.

3. Open the Properties Window of the corresponding junction and accept the pipe connectivity determination. In most junctions, this pipe connectivity determination will be located in the upper right hand corner of the Properties Window. In a few junctions, such as the [Assigned Pressure Junction](#) and [Reservoir Junction](#), the pipe connectivity will be shown under the tab containing Loss Coefficients. You can do this as you enter the junction's data in its Properties Window.

If the model seems to not be connected properly, you can also use the [Extended Model Check](#) feature on the View menu.

When you accept AFT Fathom's connectivity interpretation, the connection advances from a *graphical* connection to a *model* connection.

The model connectivity you establish on the Workspace is retained only as long as you maintain the graphical objects in their current visual relationship to each other. If you move a pipe or a junction and break the graphical connection, AFT Fathom disconnects the two objects. However, if you move a group of connected objects, their model connectivity is maintained.

Related Topics

Pipes	214
Creating Pipes - the Pipe Drawing Tool	197
Joining or Splitting Pipes	199
Moving Objects	204
Editing Objects	205
Defining Objects	206
Highlight Required Input	208
Inspecting Objects	209
Notes	210
Status	211
Junctions	248
Morphing Junctions	206
Moving Pipe and Junction Labels	212

Moving Objects

Objects on the [Workspace](#) can be moved individually or as groups. Clicking a Workspace object selects it. Pipe objects change color when selected. Junction objects are outlined in red when selected. These colors are configurable in the Workspace section of [User Options \(Tools menu\)](#) .

To move an object, select the object, drag it within the Workspace, and drop it in the desired location. When an object is dragged beyond the existing Workspace area, the Workspace is expanded accordingly.

A pipe object can be stretched by grabbing the handles (small red squares) at the pipe endpoints and moving an endpoint to a new location. To make these handles visible, select the pipe you want to move.

There are numerous ways to [select a group of objects](#) for movement or other operations.

To prevent accidental movement of objects on the Workspace, lock the objects. A lock symbol appears next to locked pipes or junctions when an attempt is made to move the pipe or junction. The Lock feature

is accessed from the [Arrange Menu](#) or the lock button on the [Toolbar](#). To lock an object on the Workspace, select it and then activate the Lock feature either from the Arrange menu or from the lock button on the Toolbar. If you would like to lock multiple objects at the same time, ensure that all of the objects are selected prior to activating the Lock feature.

Keyboard Modifiers

If you hold down the CTRL key when dragging a junction, the endpoints of any connected pipes are not moved with the junction, thus breaking the graphical connection.

Related Topics

Pipes.....	214
Creating Pipes - the Pipe Drawing Tool.....	197
Joining or Splitting Pipes.....	199
Connecting Objects.....	203
Editing Objects.....	205
Defining Objects.....	206
Highlight Required Input.....	208
Inspecting Objects.....	209
Notes.....	210
Status.....	211
Junctions.....	248
Morphing Junctions.....	206
Moving Pipe and Junction Labels.....	212

Editing Objects

The [pipes and junctions](#) you place on the [Workspace](#) can be edited with the editing commands from the Edit menu or the [Toolbar](#). Objects can be cut, copied, pasted, duplicated, and deleted. These operations can be performed on individual objects or on groups. The [Edit menu](#) provides multiple levels of undo for each editing operation. Alternately, you can use the Undo button on the Toolbar. Note that not all Arrange actions can be undone.

The Copy Graphics feature on the [Edit menu](#) will copy the Workspace objects to the Windows clipboard. This image can then be pasted into other Windows applications.

Related Topics

Pipes.....	214
Creating Pipes - the Pipe Drawing Tool.....	197
Joining or Splitting Pipes.....	199
Connecting Objects.....	203
Moving Objects.....	204

Defining Objects.....	206
Highlight Required Input.....	208
Inspecting Objects.....	209
Notes.....	210
Status.....	211
Junctions.....	248
Morphing Junctions.....	206
Moving Pipe and Junction Labels.....	212

Morphing Junctions

Occasionally, you may want to change a junction's type in your model. You can do this by morphing the old junction into a new one.

If you hold down the CTRL key, select a junction from the Toolbox, and then drop it onto an existing Workspace junction, the old junction will morph into the new one. The junction type will change to the new junction type, and data that can be kept will be copied into the new junction.

Related Topics

Junctions.....	248
Connecting Objects.....	203
Moving Objects.....	204
Editing Objects.....	205
Defining Objects.....	206
Highlight Required Input.....	208
Inspecting Objects.....	209
Notes.....	210
Status.....	211
Moving Pipe and Junction Labels.....	212

Defining Objects

To specify the required property data for an object, open its properties window. This can be done in six ways:

- Double-click the object on the Workspace
- Select the object (by clicking on it once, or using the [Selection Box](#)) and press Enter
- Select the object and click the Open Pipe/Jct Window icon on the [Workspace Toolbar](#)
- Double-click the left column of the appropriate table in the [Model Data window](#) (where the object ID number is shown)
- [Jump](#) from another Properties Window
- Double-click the connected pipe or junction number in a Properties Window

Parameters that functionally describe the pipe or junction are entered in the Properties Window. Some parameters in the Properties Windows are required and others are optional. The optional parameters offer you added control over your model and the ability to obtain specialized information.

The [highlight feature](#) shows the required input in pipes and junctions.

Undefined Objects Window

You can open the [Undefined Objects window](#) to view all undefined objects and the undefined properties. This is available on the [View](#) menu.

Use Status Feature

Each pipe and junction Properties Window has a Status tab. The Status tab shows what required information for the pipe or junction is missing.

Satisfying Connectivity Requirements

In addition to entering the required information for each object in the model, you must satisfy these connectivity requirements:

- You must accept the connectivity information shown in each object's Properties Window
- Each junction must have the appropriate number of pipes connected to it. For example, a valve must have exactly two connecting pipes, whereas a reservoir may have one to twenty-five connecting pipes.
- Pipes must always be connected to two junctions, one at each pipe endpoint.

Should you ever delete any of the required information in the Properties Window or move a connected object from its connection, the object's status reverts to undefined.

For example, say you place a [heat exchanger](#) on the Workspace. A heat exchanger requires inlet and outlet piping. AFT Fathom reflects this by requiring each heat exchanger to be assigned an upstream pipe and a downstream pipe. Entering only the heat exchanger constants and setting an elevation is **necessary but not sufficient information to define the heat exchanger** according to AFT Fathom's requirements. Once you have entered the heat exchanger data and properly connected it, the heat exchanger is defined.

Show Object Status

The Show Object Status feature helps you identify which objects on the Workspace are defined and which are not. This feature can be toggled on and off by selecting it from the [View menu](#) or by clicking the floodlight button on the [Workspace toolbar](#).

Turning Show Object Status on displays the undefined junction and pipe ID numbers in the Undefined Color, which is red by default. The color can be changed through the [User Options window](#) on the [Tools menu](#). Show Object Status helps you rapidly assess where your model requires additional information.

Leaving Show Object Status always toggled on is not recommended for large models, as it will slow the Workspace graphics response. When this feature is on, every Workspace graphical operation you perform must be accompanied by a check of all pipe and junction connections. This check takes time, and could degrade your graphics performance.

Related Topics

Pipes	214
Creating Pipes - the Pipe Drawing Tool	197
Joining or Splitting Pipes	199
Connecting Objects	203
Moving Objects	204
Editing Objects	205
Highlight Required Input	208
Inspecting Objects	209
Notes	210
Status	211
Junctions	248
Morphing Junctions	206
Moving Pipe and Junction Labels	212

Related Examples

Beginner - Three Reservoir	823
----------------------------------	-----

Highlight Required Input

The highlight feature functions to identify required input data for the Properties Window for each pipe and junction. You can toggle the highlight feature on and off in the following ways:

- From the Properties Window
 - Double-click anywhere above the tab interface in the Properties Window
 - Press the F2 function key
- From the [View Menu](#)
 - Toggle the [Highlight in Pipe and Jct Windows](#) to Off

The highlighting feature may be especially useful when you are first learning to use AFT Fathom or when you are having difficulty obtaining a defined object status.

Related Topics

Pipes.....	214
Creating Pipes - the Pipe Drawing Tool.....	197
Joining or Splitting Pipes.....	199
Connecting Objects.....	203
Moving Objects.....	204
Editing Objects.....	205
Defining Objects.....	206
Inspecting Objects.....	209
Notes.....	210
Status.....	211
Junctions.....	248
Morphing Junctions.....	206
Moving Pipe and Junction Labels.....	212

Inspecting Objects

The Inspection feature displays data associated with a Workspace object in read-only fashion via a pop-up overlay (see Figure 1). Inspecting is faster than opening the Properties window if you only want to review the data and no edits need to be made. Inspecting can be done directly from the Workspace and provides a quick summary of the parameters that appear on the [Model Data Primary Window](#).

To inspect an object, position the mouse pointer on the object and hold down the right mouse button.

The Inspection feature is also available *inside* each pipe/junction's Properties window. This is useful if you want to check the input data of objects connected to the one you are viewing. The upstream/-downstream pipes or junctions are listed, and holding the right mouse button on the listed ID number will open the associated inspection overlay. For example, when working in an [Orifice Properties window](#), you may wish to quickly check the upstream and downstream pipe diameters previously defined. Holding down the right mouse button on each connected pipe ID number will show you that pipe's primary input parameters.

Judicious inspecting can save you time when assembling or debugging a model.

Note: Output data will be shown on the inspection overlay only if a valid [Output File](#) is present for the current scenario.

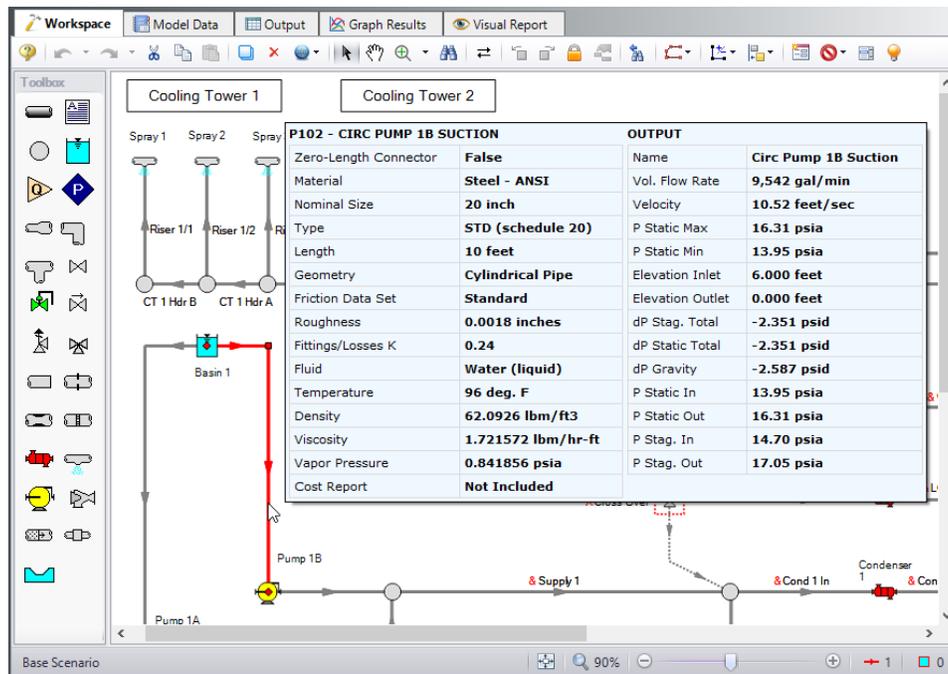


Figure 1: Holding the right mouse button on a Workspace object displays the associated data

Related Topics

Pipes.....	214
Creating Pipes - the Pipe Drawing Tool.....	197
Joining or Splitting Pipes.....	199
Connecting Objects.....	203
Moving Objects.....	204
Editing Objects.....	205
Defining Objects.....	206
Highlight Required Input.....	208
Notes.....	210
Status.....	211
Junctions.....	248
Morphing Junctions.....	206
Moving Pipe and Junction Labels.....	212

Notes

Each object can have notes associated with it. This is useful for listing assumptions, reference documents, drawings, measurements, etc.

Related Topics

Pipes.....	214
Creating Pipes - the Pipe Drawing Tool.....	197
Joining or Splitting Pipes.....	199
Connecting Objects.....	203
Moving Objects.....	204
Editing Objects.....	205
Defining Objects.....	206
Highlight Required Input.....	208
Inspecting Objects.....	209
Status.....	211
Junctions.....	248
Morphing Junctions.....	206
Moving Pipe and Junction Labels.....	212

Status

The status tab lists all undefined parameters that are required to define the pipe or junction.

Related Topics

Pipes.....	214
Creating Pipes - the Pipe Drawing Tool.....	197
Joining or Splitting Pipes.....	199
Connecting Objects.....	203
Moving Objects.....	204
Editing Objects.....	205
Defining Objects.....	206
Highlight Required Input.....	208
Inspecting Objects.....	209
Notes.....	210
Junctions.....	248
Morphing Junctions.....	206
Moving Pipe and Junction Labels.....	212

Workspace Pan

The user can use Workspace Pan mode to move around, or pan, the model. If the Workspace has been enlarged beyond the default one screen size (Workspace scrollbars will be visible), then the [Workspace](#) can be panned by activating Workspace Pan mode. There are several ways to activate Workspace Pan Mode:

- Click the Workspace Pan button (represented with a hand icon) on the toolbar
- Select the Workspace Pan option in the [View menu](#)
- If your mouse has a center button (a scroll wheel), hold the scroll wheel button down
- Hold CTRL and use the arrow keys

Once in pan mode, the cursor will change to an open hand that can be used to grab and move a point in the Workspace. The Workspace can be panned by holding down the left mouse button (if the Workspace Pan mode has been activated using the Workspace Pan button on the toolbar or the Workspace Pan option has been selected in the View menu) and moving the mouse in the desired direction, or by holding down the mouse scroll wheel (if using the scroll wheel to keep Workspace Pan mode activated) and moving the mouse in the desired direction.

To switch off the pan mode, click the mouse scroll wheel or the Workspace Pan button on the toolbar a second time. If Workspace Pan mode is maintained using the mouse scroll wheel, it will automatically turn off once you release the mouse scroll bar.

Related Topics

Moving Pipe and Junction Labels.....	212
--------------------------------------	-----

Moving Pipe and Junction Labels

Each pipe and junction on the [Workspace](#) also displays a label. A label refers to any text associated with an object - pipe numbers, names, flow rates, and so forth. The label will display a unique combination of defined display text items from each visible [Workspace Layer](#). This label can be dragged to improve model aesthetics. The relative location of the label to the object is linked so that if the object is moved, the label moves with it.

Returning Label to Default Location

To return the label to its default location, you can:

- Right-click the label for an individual object and choose Reset Label Location
- Use Arrange -> Reset Label Locations and respond to the dialog
- Click Reset Label Locations in the Label Control group of the [Layer Settings window](#) to reset all label locations

Related Topics

Junctions.....	248
Connecting Objects.....	203
Moving Objects.....	204
Editing Objects.....	205
Morphing Junctions.....	206
Defining Objects.....	206
Highlight Required Input.....	208
Inspecting Objects.....	209
Notes.....	210
Status.....	211

Workspace Pan	211
---------------------	-----

Special Conditions

Some junctions have Special Conditions which alter the normal state of the junction. For example, a valve can be closed, a pump turned off, or a relief valve opened. Pipes can also be set closed which will effectively close the pipe to flow at both ends.

The special conditions are set in one of three ways:

- By selecting the junction(s) or pipe(s) on the [Workspace](#) and choosing Special Conditions from the [Edit menu](#)
- By selecting the junction(s) or pipe(s) on the Workspace and clicking the Special Condition icon on the [Toolbar](#)
- By opening the junction's or pipe's Properties window and clicking the appropriate condition on the Optional tab

The pipes or junctions which have special conditions set are shown using a special symbol before the ID number (an "X" by default). This symbol can be customized in the [User Options](#).

When the special condition results in a section of the model being closed or "turned off", the pipes which will have zero flow are display as dotted lines. This indicates visually that they are cut off from flow; the junctions are outlined with a dotted line. The default dotted line can be changed in the User Options.

The user can specify Special Conditions for [Control Valves](#) to be either closed or fully open. Similarly, [Pumps](#) can be turned off and either allow flow through or stop the flow.

Related Topics

Optional Input for Junctions	255
Optional Input for Pipes	238
Turn Off Parts of the Model	805

Related Examples

Hot Water System	950
------------------------	-----

Pipes

This section focuses on specifying and modifying the Pipe object in the AFT Fathom Workspace. Topics include an overview of the Pipe Properties window, how to input pipe specification data, a description of the methodology used for heat exchange within pipes for AFT Fathom, and how to insert fittings and losses within the pipes.

Contents

Pipe Properties Window	214
Common Pipe Input Parameters	215
Pipe Model	216
Fittings & Losses	218
Heat Transfer in Pipes	220
Insulation for Pipes	237
Fluid Properties for Pipes	238
Optional Input for Pipes	238
Add/Remove Pipe Segment	241
Reverse Direction	242
Pressure Drop in Pipes - Detailed Discussion	243

Pipe Properties Window

Each pipe on the [Workspace](#) has engineering information associated with it. Each pipe must have an ID number, a length, a diameter, a roughness specification, heat transfer conditions (if modeling [heat transfer](#)), and two connecting junctions. If the pipe is not purely cylindrical, additional geometry information is also required.

Related Topics

Pipes	214
Common Pipe Input Parameters	215

Pipe Model.....	216
Fittings & Losses.....	218
Heat Transfer in Pipes.....	220
Insulation for Pipes.....	237
Fluid Properties for Pipes.....	238
Optional Input for Pipes.....	238
Add/Remove Pipe Segment.....	241
Reverse Direction.....	242
Pressure Drop in Pipes - Detailed Discussion.....	243
Pipe X length is shorter than the elevation change.....	789

Common Pipe Input Parameters

The common input parameters are located above the tabs in the [Pipe Properties window](#).

Number

Every pipe must have a unique ID number. When new pipes are created, a pipe number is automatically allocated.

The pipe number can be changed by the user to any unique value greater than zero and less than 100,000. Pipe numbers are only identifiers and have no affect on the model behavior.

Name

A name can be assigned to each pipe for reference purposes. The default name is simply "Pipe". Names do not need to be unique. The name can be shown on the Workspace and in the Output window.

Copy Data From Pipe...

The Copy Data From Pipe dropdown list allows you to copy all or some of an existing pipe's properties to the current pipe. Selecting an existing pipe will prompt the user on what parameters should be changed. The Copy Previous button functions similarly, copying parameters from the previously edited pipe.

In a child [scenario](#), Copy Data From Pipe lists "Parent Pipe Data," allowing the user to copy all or some parameters from the parent pipe.

Connected Junctions

All pipe objects have a [reference positive direction](#), as indicated on the Workspace by an arrow in the center of the pipe. The upstream and downstream junctions are identified with this direction. The connected junctions area displays these junctions. Every pipe must be [connected to two junctions](#) in order to be completely defined.

Related Topics

Pipes.....	214
Pipe Properties Window.....	214
Pipe Model.....	216
Fittings & Losses.....	218
Heat Transfer in Pipes.....	220
Insulation for Pipes.....	237
Fluid Properties for Pipes.....	238
Optional Input for Pipes.....	238
Add/Remove Pipe Segment.....	241
Reverse Direction.....	242
Pressure Drop in Pipes - Detailed Discussion.....	243

Pipe Model
Size

The size section of the Pipe Model tab specifies the pipe geometry.

- **Pipe Material**
 - **User Specified** - If selected, the geometry of the pipe must be directly specified.
 - **Library Material** - Many common pipe materials are included in the standard [pipe material library](#). These and any other accessible library materials can be selected. When a library material is selected, the parameters defining the geometry cannot be modified.
- **Pipe Geometry** - Several pipe geometries can be modeled. The critical value for flow calculations is hydraulic diameter, based on cross-sectional area A_c and wetted perimeter P_w :

$$D_H = \frac{4A_c}{P_w}$$

- **Cylindrical Annulus** - Flow in an annulus or "ring" pattern. This requires both an outer and inner diameter to calculate hydraulic diameter.
- **Cylindrical Pipe (default)** - A typical pipe. The hydraulic diameter and inner diameter are identical
- **Helical Tube** - Requires hydraulic diameter and coil diameter. Specification of this geometry affects the friction model pressure loss.
- **Noncylindrical Pipe** - Requires flow area (cross-sectional area) and wetted perimeter to calculate hydraulic diameter.
- **Rectangular Pipe** - Requires width and height to calculate hydraulic diameter.

- **Size** - Select from a list of defined nominal sizes for the selected Pipe Material. Only available for library materials.
- **Type** - Select from a list of defined types (classes/schedules) for the selected Pipe Material. Only available for library materials.
- **Geometric Specification** - Several required fields that change depending on the selected Pipe Geometry. Only editable for User Specified Pipe Material.
- **ID Reduction (Scaling)** - Reduction in hydraulic diameter due to scaling can be accounted for by entering a percent reduction in this field. Zero represents no reduction.

Length

The Length is defined as the distance between the two connecting junctions. This length has no relationship to the visual length of the pipe drawn on the [Workspace](#).

Model As Zero-Length Connector can be checked to turn the pipe into a lossless connection between two junctions, such as for two components that are directly welded together. Zero-Length Connectors (ZLCs) are denoted by a red C icon next to the pipe name on the Workspace. Intermediate branching flow is not allowed between two ZLCs.

ZLCs are hidden by default in the Model Data and Output. Generally the output for ZLCs is not of interest because they will have the same inlet and outlet properties.

You can enable ZLCs to be shown in the Model Data by opening the Model Data Control, going to the Show Selected Pipes/Junctions tab, and selecting the Show Zero-Length Connectors checkbox under the list of pipes. The same checkbox is present in the Output Control on the Show Pipes/Jcts tab. You will then need to click on the pipe numbers in the list to make them visible in the Model Data or Output, or you can use the All button to select all pipes in the model.

Friction Model

Multiple [friction data sets](#) can be compiled for each library pipe material. These different data sets are then made available in the Data Set drop down list. For AFT standard materials, the friction data set is generally "Standard."

The Friction Model can be modified for any individual pipe, and the default friction model can be set in [Parameter Options](#). The Friction Model for a library material can be modified by selecting the User Specified radio button in Data Set, and must be directly specified for User Specified materials.

These models apply to Newtonian viscosity models. If a [non-Newtonian](#) model is selected in Analysis Setup, the below friction models may not be used.

- **Absolute roughness (default)** - Absolute average roughness height. Values of pipe roughness can be found in many pipe handbooks or from manufacturer's data.
- **Relative roughness** - Relative roughness is the absolute roughness divided by the pipe diameter.
- **Hydraulically smooth** - Hydraulically smooth implies that roughness is negligible. This is not the same as frictionless.
- **Explicit Friction Factor** - Friction factor for the pipe, you can enter the value explicitly.

- **Frictionless** - For troubleshooting purposes, it is occasionally useful to model a pipe as having no friction. There are limitations to where such a pipe can be located in your model.
- **Hazen-Williams** - A method using an empirical factor to relate flow and pressure drop, common in water distribution and NFPA fire protection.
- **MIT Equation** - A model appropriate for crude oil.
- **Miller Turbulent** - A model appropriate for light hydrocarbons.

For detailed information on how these Friction Models are defined, see [Pressure Drop in Pipes - Detailed Discussion](#).

Related Topics

Pipes.....	214
Pipe Properties Window.....	214
Common Pipe Input Parameters.....	215
Fittings & Losses.....	218
Heat Transfer in Pipes.....	220
Insulation for Pipes.....	237
Fluid Properties for Pipes.....	238
Optional Input for Pipes.....	238
Add/Remove Pipe Segment.....	241
Reverse Direction.....	242
Pressure Drop in Pipes - Detailed Discussion.....	243

Fittings & Losses

Fittings & Losses allows the user to define additional losses directly in a pipe. This is useful to account for pressure losses without modeling large numbers of elbows or similar fittings. To specify losses, open the Pipe Properties window, click the Fittings & Losses tab, then click Specify Fittings & Losses to open the Pipe Fittings & Losses window. An Additional K Factor loss can also be user specified.

Pipe Fittings & Losses

The Fittings & Losses library contains many standard loss values. Because many of these values are dependent on pipe diameter, a pipe diameter must be specified to open the Pipe Fittings & Losses window.

Losses are divided into several categories represented by tabs. To add fittings to the pipe, select the type, and adjust the quantity column next to the desired fitting.

After accepting the changes by clicking OK, a summary of the pipe's Fittings & Losses will be displayed in the Pipe Properties window.

Note: Area Change fittings do not change the size of the pipe, they only represent the loss factor based on the pipe diameter and the area ratio of the loss. When including area changes, notice the illustration of how the loss is defined, where A_{pipe} is the area of the pipe and A_s is the area of the loss. These area changes are useful to model fittings between a pipe and another junction like a pump.

If you are using [Equivalent Lengths](#), the table displayed will be a simpler form that allows the addition of specific bends and valves (the equivalent lengths displayed here are taken from [NFPA13](#)). You can also simply type in additional equivalent length.

The bottom of the table will show a Total Applied value which is a count of the number of losses that have been specified for the currently selected tab.

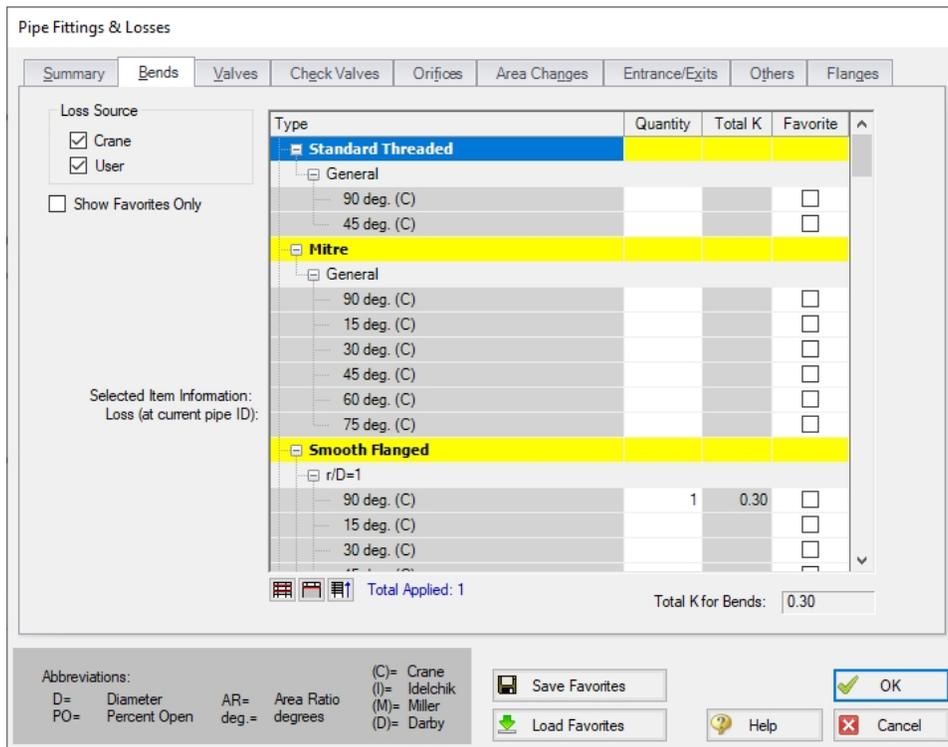


Figure 1: Pipe Fittings & Losses window

Fittings & Losses Workspace Symbol

When a pipe has any loss value assigned through the Fittings & Losses tab, a symbol ([default of &](#)) is added before the pipe number.

Favorite Fittings & Losses

In the Pipe Fittings & Losses window, there is a column labeled Favorite for each of the loss types. Check the box for favorite to add it to the list of favorite fittings and losses. On the left side of the window there are several options to filter the list. For example, one may only want to see Crane fittings. In which case, Crane should be checked, and all others unchecked on the left hand side of the window. If a list of only favorite fittings is desired, check the box to "Show Favorites Only".

Related Topics

Pipes	214
Pipe Properties Window.....	214
Common Pipe Input Parameters.....	215
Pipe Model.....	216
Heat Transfer in Pipes.....	220
Insulation for Pipes.....	237
Fluid Properties for Pipes.....	238
Optional Input for Pipes.....	238
Add/Remove Pipe Segment.....	241
Reverse Direction.....	242
Pressure Drop in Pipes - Detailed Discussion.....	243
Fittings & Losses Library.....	717
Equivalent Length.....	267

Related Examples

Sizing a Pump	848
---------------------	-----

Heat Transfer in Pipes

The Heat Transfer tab is only visible when [heat transfer](#) is being modeled. Heat transfer modeling is enabled in [Analysis Setup](#) on the [Heat Transfer/Variable Fluids panel](#).

Each pipe responds thermally to the environment outside of the pipe. AFT Fathom offers several models that allow you to model the heat transfer characteristics of a pipe:

- [Adiabatic \(Perfectly Insulated\)](#) - An adiabatic pipe is one that has perfect insulation and thus no heat transfer takes place. This model differs from Isothermal in that the temperature into the pipe is taken from the upstream junction. This is the default model. [Insulation data](#) can be entered but is not used for thermal purposes.
- [Isothermal](#) - The Isothermal model maintains the pipe at the specified temperature. [Insulation data](#) can be entered but is not used for thermal purposes.
- [Convective Heat Transfer](#) - This model allows you to specify heat transfer data for the internal fluid, pipe wall, three layers of insulation and external convection.

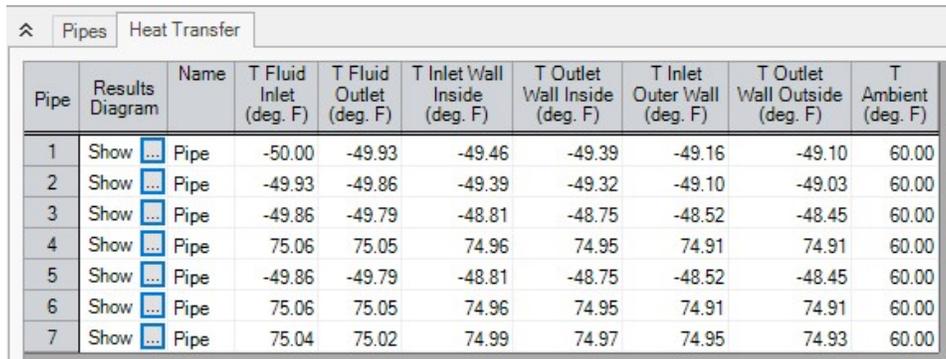
Pipes

- [Convective Heat Transfer and Heat Flux](#) - This model allows you to model convective heat transfer while also specifying a constant heat flux.
- [Convective Heat Transfer with Heat Tracing](#) - This represents adding some type of heating layer onto the pipe's outer surface. The amount of heating may be entered as a constant heat flux or by providing information on the power and number of wire turns. Insulation layers can then be added on top of the heat tracing (heat flux source).
- [Constant Heat Flux](#) - The constant heat flux model is useful in cases where the temperature difference between the fluid and the ambient environment is extremely large or in cases where a large radiant heat flux exists. Insulation data can be entered but is not used for thermal purposes.
- [Constant Heat Rate](#) – This model allows you to specify a constant heat rate into or out of the pipe.
- [Buried Pipe](#) - This model allows you to specify that the pipe is buried in a thermally conductive medium.

Note that similar to flow rate definition, heat flow is positive when flowing into the pipe, and negative when flowing out.

Viewing Calculated Thermal Results in Output

A table is displayed in the Output window Pipe area showing the temperature distribution in the pipe and insulation.



Pipe	Results Diagram	Name	T Fluid Inlet (deg. F)	T Fluid Outlet (deg. F)	T Inlet Wall Inside (deg. F)	T Outlet Wall Inside (deg. F)	T Inlet Outer Wall (deg. F)	T Outlet Wall Outside (deg. F)	T Ambient (deg. F)
1	Show ...	Pipe	-50.00	-49.93	-49.46	-49.39	-49.16	-49.10	60.00
2	Show ...	Pipe	-49.93	-49.86	-49.39	-49.32	-49.10	-49.03	60.00
3	Show ...	Pipe	-49.86	-49.79	-48.81	-48.75	-48.52	-48.45	60.00
4	Show ...	Pipe	75.06	75.05	74.96	74.95	74.91	74.91	60.00
5	Show ...	Pipe	-49.86	-49.79	-48.81	-48.75	-48.52	-48.45	60.00
6	Show ...	Pipe	75.06	75.05	74.96	74.95	74.91	74.91	60.00
7	Show ...	Pipe	75.04	75.02	74.99	74.97	74.95	74.93	60.00

Figure 1: Thermal Data table in Output window Pipe area shows temperature distribution in pipe

Clicking the button next to Show brings up a Results Diagram for that particular pipe:

Constant Heat Transfer

There are several thermal models for pipes in AFT Fathom which fix the thermal behavior of the pipe to some constant value:

- **Adiabatic (Perfectly Insulated)** - The heat transfer between the pipe and the environment is fixed to zero. The entire length of the pipe will have a stagnation enthalpy equal to that of the upstream junction. The actual fluid temperature depends on the system.
- **Isothermal** - The fluid temperature is set to a specified value. Heat transfer is still modeled in this case - instead of varying the fluid temperature, the heat rate into or out of the pipe is found that satisfies the isothermal constraint.
- **Constant Heat Flux** - A constant heat flux (heat rate per area) is applied to the pipe. Positive values indicate heat entering the pipe. Flux is often a more readily available and flexible value for some conditions, such as radiation heat transfer. For this model, the heat flux is applied to the area defined by the ID of the pipe. To model a heat flux underneath insulation, use the Convective Heat Transfer with Heat Tracing model.
- **Constant Heat Rate** - It may also be known that a certain pipe transfers (or needs to transfer) a specific heat rate, regardless of the size of the pipe. This model allows the specification of a constant energy per time into or out of the pipe.

The above models follow a common format on the Pipe's Heat Transfer tab, as shown below. There is only one required value input unless insulation is added. One layer of insulation (external or internal) can be added, with conductivity values from a library or User Specified in this window.

Note: The Heat Transfer tab will not be present unless heat transfer has been enabled from the [Heat Transfer/Variable Fluids panel](#) of the Fluid Properties group in the Analysis Setup window.

The screenshot shows the 'Pipe Properties' dialog box with the 'Heat Transfer' tab selected. The 'Heat Transfer Model' is set to 'Constant Heat Flux'. The 'External Heat Flux' is set to '1' with units of 'Btu/s-ft2'. The 'Insulation Type' is set to '(None)'. Other fields include 'Number' (8), 'Name' (Pipe), 'Upstream Junction' (None), 'Downstream Junction' (None), 'Copy Data From Pipe...', 'Copy Previous...', 'OK', 'Cancel', 'Jump...', and 'Help' buttons. The 'Heat Transfer' tab is highlighted with a red square.

Figure 1: Constant Heat Flux

Related Topics

Heat Transfer in Pipes..... 220

Convective Heat Transfer.....	224
Heat Tracing Flux based on Wire Turns and Length.....	227
Buried Pipe Heat Transfer.....	227
Heat Transfer in Pipes - Detailed Discussion.....	231

Related Examples

Heat Transfer in a Pipe.....	861
------------------------------	-----

Convective Heat Transfer

There are 3 convective based models for pipe heat transfer:

- **Convective Heat Transfer** - Represents a pipe thermally affected solely by convection.
- **Convective Heat Transfer and Heat Flux** - Behaves the same as the Convective Heat Transfer model, but also includes the effect of an external heat flux such as radiation.
- **Convective Heat Transfer with Heat Tracing** - Similar to the Convective Heat Transfer and Heat Flux model, but instead of applying the flux on the outermost surface of the pipe/insulation, the flux can be applied directly to the pipe. As the name suggests, this represents a device such as a heated wire. If the flux is not known directly, it is easy to determine [based on Wire Turns and Length](#).

Defining the Convective Model

When one of the convective heat transfer models is selected, there are a number of items that must be defined:

- **Resistance Geometry** - Radial or Linear.
- **Number of Insulation Layers** - There can be up to three layers of insulation defined, in any combination of internal or external.
- **External Heat Flux** - Only available if Convective Heat Transfer and Heat Flux is selected. A constant flux applied to the outermost layer of resistance.
- **Heat Tracing on Pipe (Flux)** - Only available if Convective Heat Transfer with Heat Tracing is selected. A constant flux is applied at the outer wall of the pipe.
- **Ambient Conditions** - The temperature and velocity of the fluid around the pipe. The fluid can be either air or water. These values are used for calculating the convection coefficient if correlation is selected as the convection Data Source. This correlation can be used for pipes over which boundary layers develop freely, free of constraints imposed by other surfaces. All of the external fluid properties are evaluated at the film temperature using data from the [NIST REFPROP fluid library](#).
- **Resistance Table** - The table at the bottom of the window defines all of the resistance layers used in the thermal model.

- **Rows**
 - **Units** - This special row defines the units of any value in the column below a cell.
 - **Fluid Internal** - The fluid itself has a convection coefficient that affects the transfer of energy to the pipe wall. By default, the Gnielinski Correlation is used to determine this value. Alternatively, the Dittus-Boelter Correlation can be used, or a User Specified value can be entered.
 - **Pipe Wall** - The wall of the pipe has a thermal conductivity that affects the heat transfer. The built-in pipe materials have thermal conductivity values defined, and this value is automatically selected. If a User Specified pipe is used, the conductivity will need to be entered.
 - **Insulation** - There can be several insulation rows, depending on the selection of Number of Insulation Layers. The Insulation can be selected from the Insulation library, in which case the conductivity is read from the library. User Specified insulation requires this value to be defined.
 - **External** - Convection occurring external to the pipe and any insulation layers. By default the Convection Coefficient is calculated with the Churchill-Bernstein Correlation. The Correlation can also be set to Free-Horizontal or Free-Vertical which requires only an ambient temperature, and no fluid velocity. The Free Horizontal and Free Vertical orientations are in reference to the orientation of the pipes. If the convection coefficient is known, a value can be entered by the user.
- **Columns**
 - **Apply** - This can allow enabling/disabling of including certain layers in the calculation. This is useful for comparing states or scenarios without redefining the thermal model.
 - **Conductivity/Convection Data Source** - Specifies from where relevant information is attained. This can be a Library, Correlation, or User Specified.
 - **Conductivity** - Applies to the Pipe Wall and Insulation. If Data Source is a Library, the particular insulation must be selected. If User Specified, the conductivity must be entered. Note that obtaining the thermal conductivity from an insulation or pipe material library permits temperature dependent thermal conductivity to be employed. With this option, AFT Fathom dynamically selects the insulation thermal conductivity during model solving based on the local temperatures that exist. Conversely, user specified values are always constant.
 - **Thickness** - Applies to Pipe Wall and Insulation. If a Library is used for the Pipe Material, this value is automatically populated. All insulation layers require a thickness. If a Pipe Material is chosen on the Pipe Model tab, and that material has insulation data sets associated with it, an option is provided in the Conductivity/Convection Data Source column to select that insulation data set. The insulation data sets include the insulation thickness, and value is displayed in the thickness column.
 - **HT Area Ratio** - Multiplier for the value of surface area used in the heat transfer calculations (default value of 1). In general, this represents any "extra" heat transfer area that may exist. For example, cooling fins on the outside of the pipe could be modeled by entering a value larger than 1 on the External resistance row.

Pipes

- **Convection Coefficient** - Applies to Fluid Internal and External. Both values can be either Correlation or User Specified.

Resistance Type	Apply	Conductivity/Convection Data Source	Conductivity	Thickness	HT Area Ratio	Convection Coefficient
Units			Btu/hr-ft-R	inches		Btu/hr-ft ² -R
Fluid Internal	<input checked="" type="checkbox"/>	Correlation			1	Gnielinski
Pipe Wall	<input checked="" type="checkbox"/>	Material Library	From Library	0.049	1	
Insulation #1 (external)	<input checked="" type="checkbox"/>	Insulation Library	Fiberglass		1	
External	<input checked="" type="checkbox"/>	Correlation			1	Forced (Churchill-Bernstein)

Figure 1: Convective Heat Transfer Layout

Modeling a Constant Wall Temperature

Assign an ambient temperature and disable the External Resistance by unchecking the box in the Apply column. This will force the outside wall of the pipe to be equal to the ambient temperature. Note that this differs from isothermal in that the fluid temperature can still change - only the outside wall of the pipe has a fixed temperature.

Related Topics

Heat Transfer in Pipes.....	220
Constant Heat Transfer.....	223
Heat Tracing Flux based on Wire Turns and Length.....	227
Buried Pipe Heat Transfer.....	227
Heat Transfer in Pipes - Detailed Discussion.....	231

Related Examples

Heat Transfer in a Pipe.....	861
Beginner - Heat Transfer in a Pipe - GSC.....	994

Buried Pipe Heat Transfer

Buried Pipe Heat Transfer follows a form very similar to the [Convective Heat Transfer](#) models.

Resistance Type	Apply	Conductivity/Convection Data Source	Conductivity	Thickness	HT Area Ratio	Convection Coefficient
Units			Btu/hr-ft-R	inches		Btu/hr-ft ² -R
Fluid Internal	<input checked="" type="checkbox"/>	Correlation			1	Gnielinski
Pipe Wall	<input checked="" type="checkbox"/>	Material Library	From Library	0.049	1	
Soil	<input checked="" type="checkbox"/>	User Specified			1	

Figure 1: Buried Pipe Window

The differences between convective heat transfer and buried pipe heat transfer are:

- The Ambient Conditions are now the Soil Surface Temperature and Buried Pipe Depth. Specifically, the Soil Surface Temperature asks for the temperature of the soil at the ground surface (as opposed to the ambient air temperature), while the Buried Pipe Depth asks for the distance between the ground surface and pipe centerline.
- The External Resistance Type is replaced with Soil. This requires a User Specified value of conductivity for the soil

Related Topics

Heat Transfer in Pipes.....	220
Constant Heat Transfer.....	223
Convective Heat Transfer.....	224
Heat Tracing Flux based on Wire Turns and Length.....	227
Heat Transfer in Pipes - Detailed Discussion.....	231

Related Examples

Heat Transfer in a Pipe.....	861
------------------------------	-----

Heat Tracing Flux based on Wire Turns and Length

The [Convective Heat Transfer with Heat Tracing](#) model requires the user enter additional information to represent the thermal effect of a heat tracing.

Pipes

Fathom allows users the ability to model Heat Tracing by either entering the heat flux directly, or by providing information on the number of turns and power per unit length of wire, which Fathom will use to determine the heat flux as described below.

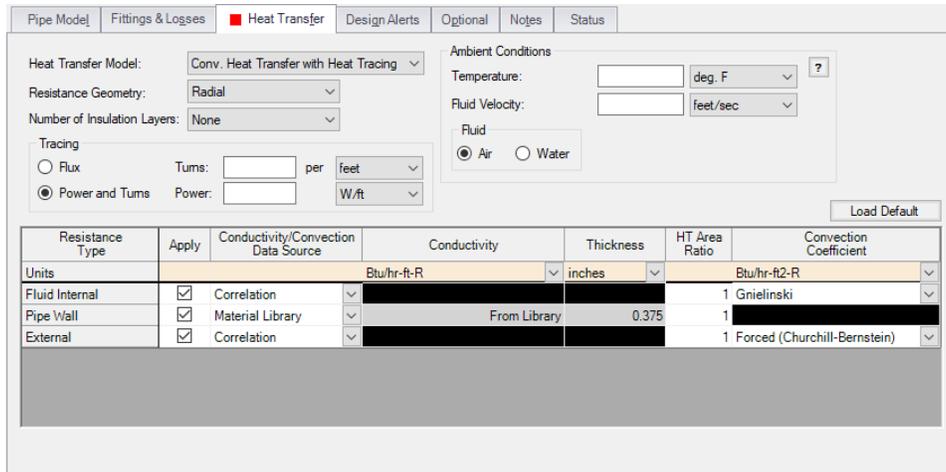


Figure 1: Pipe Properties window with the Conv. Heat Transfer with Heat Tracing model enabled

Power and Turns - Detailed Discussion

When you select the option to model Heat Tracing based off a heated wire, Fathom models that heat flux based on the theory described below.

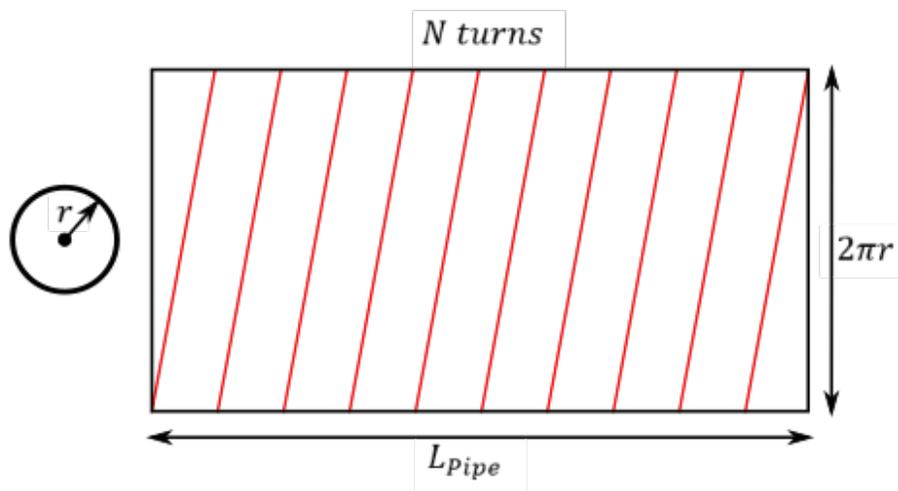


Figure 2: Wire tracing geometry

Length of Wire

First, it may be necessary to determine the length of the wire itself, based on a number of turns.

- N - total number of turns on the pipe
- n - number of turns per unit length
- L_{Pipe} - total length of the pipe
- r - external radius of the pipe
- L_{Wire} - length of the wire
- L_{Turn} - length of one wire turn

The above figure shows a pipe with heat tracing that has been "split open" and laid into a flat rectangle to make these equations more intuitive.

The total turns and turns per length are related by the pipe length:

$$N = nL_{Pipe}$$

Each turn of the wire creates a right triangle on the "flattened" pipe, so we can find the length of each segment by the Pythagorean theorem.

$$L_{Turn}^2 = \left(\frac{L_{Pipe}}{N}\right)^2 + (2\pi r)^2$$

The total length of the wire is related to the number of turns and the segment length.

$$L_{Wire} = NL_{Turn}$$

Incorporating this relationship and rearranging:

$$L_{Wire}^2 = L_{Pipe}^2 + (2\pi r N)^2$$

Using the number of turns per unit length:

$$L_{Wire}^2 = L_{Pipe}^2 + (2\pi r n L_{Pipe})^2 = L_{Pipe}^2 (1 + (2\pi r n)^2)$$

or:

$$L_{Wire} = L_{Pipe} \sqrt{1 + (2\pi r n)^2}$$

Flux of Wire

With the length of the wire found, the flux based on power per length can be found.

- P_{Total} - total power usage of the wire, along the entire pipe length
- p - power per unit length
- Q'' - heat flux
- A - area heat flux is applied to

The heat flux is defined as the heat delivered per area. Here, we assume that the "total power" used by the heat trace is equivalent to the total heat delivered. Therefore:

$$Q'' = \frac{Q}{A} = \frac{P_{Total}}{A}$$

The total power is equal to the power per length times the length, and area is related to pipe geometry:

$$Q'' = \frac{pL_{Wire}}{2\pi r L_{Pipe}}$$

Substituting in for L_{Wire} :

$$Q'' = \frac{pL_{Pipe} \sqrt{1 + (2\pi r n)^2}}{2\pi r L_{Pipe}}$$

Simplifying:

$$Q'' = p \sqrt{\frac{1}{(2\pi r)^2} + n^2}$$

Related Topics

Heat Transfer in Pipes.....	220
Constant Heat Transfer.....	223
Convective Heat Transfer.....	224
Buried Pipe Heat Transfer.....	227
Heat Transfer in Pipes - Detailed Discussion.....	231

Related Examples

Heat Transfer in Pipes - Detailed Discussion

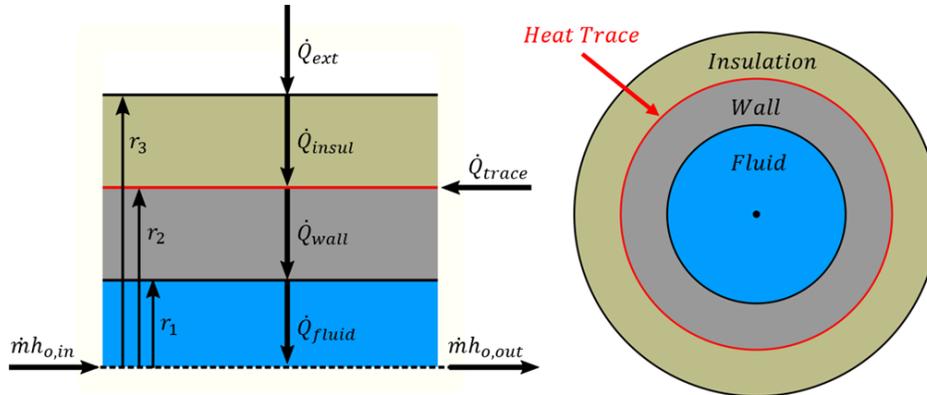


Figure 1: Diagram of a pipe with Heat Tracing and Insulation

There are many possible combinations of heat transfer that can be defined in AFT Fathom. In any of these models there are various heat flows that can be considered.

Figure 1 shows the heat rates for one of the more complex cases – convective heating with a heat trace on the outer face of the pipe wall. Because this is a steady-state problem, any control volume that is drawn around sub-sections or the entire section must have entering heat equal to leaving heat.

Balancing these heat rates and temperatures via iterative methods allows AFT Fathom to come to a unique solution.

Heat Rates

As a hydraulic solver, the primary heat rate of interest is \dot{Q}_{fluid} , the heat added to the fluid itself. Adding heat to the fluid has the effect of raising its temperature.

We can derive an expression for \dot{Q}_{fluid} with the first law of thermodynamics. For convenience, we use specific quantities (per unit mass)

$$e_{out} - e_{in} = q - w$$

The energy e is composed of internal energy u and kinetic energy $V^2/2$. Note that the gravity term gz is neglected from this equation to simplify the derivation.

$$e = u + \frac{V^2}{2}$$

$$u_{out} + \frac{V_{out}^2}{2} - \left(u_{in} + \frac{V_{in}^2}{2} \right) = q - w$$

The work done by the fluid is comprised of pressure work pv and shaft work w_{shaft}

$$w = (pv)_{out} - (pv)_{in} + w_{shaft}$$

Combining and rearranging

$$\left(u_{out} + (pv)_{out} + \frac{V_{out}^2}{2} \right) - \left(u_{in} + (pv)_{in} + \frac{V_{in}^2}{2} \right) = q - w_{shaft}$$

Using the definition of enthalpy

$$h = u + pv$$

$$h_{out} + \frac{V_{out}^2}{2} - h_{in} - \frac{V_{in}^2}{2} = q - w_{shaft}$$

In this analysis, we consider that the velocity through the control volume remains constant. In reality, changing temperature will cause the density of the fluid to change, and thus the velocity. For this reason, pipes in AFT Fathom are thermally sectioned. This means that certain properties are considered constant throughout the section as an approximation – having several sections in a pipe helps increase the accuracy of this approximation.

There is also no shaft work being done on the fluid.

$$\Delta h = q$$

We further assume that this can be treated thermodynamically as a constant pressure process. That is, c_p is constant in the thermal section, like density. With this assumption we can use the relationship

$$\Delta h = c_p \Delta T_{fluid}$$

To find that

$$q = c_p \Delta T_{fluid}$$

Rather than using specific quantities, it is useful to relate heat rate and mass flow. Multiply both sides by \dot{m}

$$\dot{Q}_{fluid} = \dot{m}c_p\Delta T_{fluid}$$

The other heat rates in the system can be related to \dot{Q}_{fluid} and other user defined values like \dot{Q}_{flux} or \dot{Q}_{trace} by examining various control volumes and surfaces:

$$\begin{aligned} \dot{Q}_{fluid} &= \dot{Q}_{trace} + \dot{Q}_{ext} && CV \text{ around wall \& insulation} \\ \dot{Q}_{wall} &= \dot{Q}_{fluid} && CS \text{ at inside of wall} \\ \dot{Q}_{ext} &= \dot{Q}_{insul} && CS \text{ at outside of insulation} \end{aligned}$$

We now have a system of equations that allow us to determine all the heat rates in the system given a mass flow (determined hydraulically), a specific heat (defined versus temperature) and a change in temperature. However, we do not know what this change in temperature is. To find it, we need to relate the surface temperatures to the heat rates.

Surface Temperatures and Resistance

Thermal resistances are useful because they directly relate the heat rate between two points with the temperature between those points.

$$R_{layer} = \frac{\Delta T_{Layer}}{\dot{Q}}$$

The value of resistance depends on the mechanism of heat transfer

$$R_{convection} = \frac{1}{hA}$$

$$R_{conduction} = \frac{\Delta r}{kA}$$

$$R_{soil} = \frac{1}{2\pi kL} \ln \left[\frac{z_{depth}}{r_{pipe}} + \sqrt{\left(\frac{z_{depth}}{r_{pipe}}\right)^2 - 1} \right]$$

For conduction, the area and thus resistance changes with increasing r , so an integration must be done

$$\int_0^R dR_{conduction} = \int_{r_1}^{r_2} \frac{dr}{kA_{eff}} = \int_{r_1}^{r_2} \frac{dr}{kA_{ratio}2\pi rL}$$

$$R_{conduction} = \frac{\ln(r_2/r_1)}{kA_{ratio}2\pi L}$$

These resistances can all be calculated with user-defined data. AFT Fathom helps simplify this by offering several correlations that can be used to determine the heat transfer coefficients in the above equations from more readily available parameters.

With resistances and \dot{Q}_{fluid} , all the surface temperatures can be determined. By bounding the system with a given $T_{ambient}$ we can work outside in to find the required T_{fluid} to obtain those heat rates.

$$\frac{T_{ambient} - T_3}{R_{convection}} = \dot{Q}_{ext}$$

$$\frac{T_3 - T_2}{R_{insulation}} = \dot{Q}_{insulation}$$

$$\frac{T_2 - T_1}{R_{wall}} = \dot{Q}_{wall}$$

$$\frac{T_1 - T_{fluid}}{R_{fluid}} = \dot{Q}_{fluid}$$

If the result for T_{fluid} matches the values used for ΔT_{fluid} used for determining \dot{Q}_{fluid} , the correct solution has been found. If they are different, ΔT_{fluid} needs to be adjusted and the process repeated until reaching a stable solution.

Overall Heat Transfer

Wherever there are sequential thermal losses, it is convenient to combine these losses. This is straightforward:

$$R_{total} = \sum R_{layer}$$

Additionally, an overall heat transfer coefficient U can be defined which behaves similarly to h or k

$$R_{total} = \frac{1}{UA}$$

Using the heat rate and resistance relationship, as well as the fact the heat transfer is into the fluid

$$\dot{Q}_{total} = \frac{\Delta T_{radial}}{R_{total}} = \dot{Q}_{fluid}$$

$$UA(T_{external} - T_{fluid}) = \dot{m}c_p \Delta T_{fluid}$$

Transforming the equation into a differential form on radius and length

$$UP(T_{external} - T_{fluid})dx = \dot{m}c_p dT_{fluid}$$

Rearranging and integrating

$$\int_0^L \frac{UP}{\dot{m}c_p} dx = \int_{T_{in}}^{T_{out}} \frac{1}{T_{external} - T_{fluid}} dT_{fluid}$$

$$\frac{UPL}{\dot{m}c_p} = -\ln \left(\frac{T_{external} - T_{out}}{T_{external} - T_{in}} \right)$$

We arrive at an expression relating the temperature change along the pipe and temperature change through the pipe:

$$\boxed{e^{\frac{-UPL}{\dot{m}c_p}} = \frac{T_{external} - T_{out}}{T_{external} - T_{in}}}$$

Correlations

Thermal resistances depend on the conduction and convection coefficients k and h which can be directly specified or calculated via a correlation.

The convection coefficient for internal fluid convection can be calculated with the Gnielinski Correlation or the Dittus-Boelter Correlation found in [Incropera](#). These correlations determine the Nusselt number of the fluid – a ratio of convective to conductive heat transfer. Conductive properties are required input – with a Nusselt number and conductivity defined, the convection coefficient can be calculated.

$$Nu = \frac{\text{Convection}}{\text{Conduction}} = \frac{h}{k/L}$$

These correlations also make use of the Prandtl number, a ratio of viscous diffusion to thermal diffusion.

$$Pr = \frac{\text{Viscous diffusion}}{\text{Thermal diffusion}} = \frac{\mu/\rho}{k/(c_p\rho)} = \frac{c_p\mu}{k}$$

Gnielinski

$$Nu = \frac{(f/8)(Re - 1000)Pr}{1 + 12.7(f/8)^{1/2}(Pr^{2/3} - 1)} \quad \text{valid for } \left(\begin{array}{l} 0.5 \leq Pr \leq 2000 \\ 3000 \leq Re \leq 5 * 10^6 \end{array} \right)$$

Dittus-Boelter

$$Nu = 0.023 Re_d^{4/5} Pr^n \quad \text{where } \left(\begin{array}{l} n = 0.4 \text{ for heating} \\ n = 0.3 \text{ for cooling} \end{array} \right) \quad \text{valid for } \left(\begin{array}{l} 0.6 \leq Pr \leq 160 \\ Re \geq 10^4 \end{array} \right)$$

External convection behavior differs from the above. AFT Fathom provides the Churchill-Bernstein correlation for determining a convection coefficient from ambient temperature and fluid velocity. This correlation can be used for pipes over which boundary layers develop freely, free of constraints imposed by other surfaces. All of the external fluid properties are evaluated at the film temperature.

$$\overline{Nu} = 0.3 + \frac{0.62Re^{1/2}Pr^{1/3}}{\left[1 + \left(\frac{0.4}{Pr}\right)^{2/3}\right]^{1/4}} \left[1 + \left(\frac{Re}{282,000}\right)^{5/8}\right]^{4/5} \quad \text{valid for } (Pr * Re \geq 0.2)$$

Buried Pipe Heat Transfer

[Buried Pipe heat transfer](#) is modeled by solving directly for the thermal resistance of the soil layer. AFT Fathom uses a model developed by [Eckert and Drake](#) to directly calculate the thermal resistance of soil on top of a pipe for use in the overall thermal resistance network:

$$R_{soil} = \frac{1}{2\pi kL} \ln \left[\frac{z_{depth}}{r_{pipe}} + \sqrt{\left(\frac{z_{depth}}{r_{pipe}}\right)^2 - 1} \right]$$

In this model, z is the distance between the ground surface and the pipe centerline, while r is the outside radius of the pipe.

Related Topics

Heat Transfer in Pipes.....	220
Constant Heat Transfer.....	223
Convective Heat Transfer.....	224
Heat Tracing Flux based on Wire Turns and Length.....	227
Buried Pipe Heat Transfer.....	227

Related Examples

Heat Transfer in a Pipe.....	861
------------------------------	-----

Insulation for Pipes

In general, insulation data can be entered either on the Insulation tab or Heat Transfer tab. Only one of these tabs displays at a time. Insulation input depends on whether or not [heat transfer](#) is being modeled, and is an optional input. Insulation data is from the [Nayyar Piping Handbook 2000](#) pages B.325-B.344.

If Heat Transfer Is Not Modeled

If heat transfer is not modeled, then a separate Insulation tab appears on the [Pipe Properties Window](#). The insulation will have no thermal effect.

- **Insulation on Outside of Pipe or Duct** - If the insulation is on the outside of the pipe or duct, it will have no hydraulic effect, and will only affect cost calculations if they are enabled.
- **Insulation on Inside of Pipe or Duct** - If the insulation is on the inside of the pipe or duct, it will reduce the flow area and change the pressure drop calculations. AFT Fathom accounts for this diameter reduction in such cases. The user must enter the insulation friction characteristics.

If Heat Transfer Is Modeled

When heat transfer is modeled, insulation input occurs on the [Heat Transfer](#) tab. There is no Insulation tab in this case.

- **Convective Heat Transfer Not Used** - If a heat transfer model other than convective heat transfer is selected, the effect of insulation is the same as discussed above for the non-heat transfer case.
- **Convective Heat Transfer Is Used** - If convective heat transfer is selected, the effect of insulation is the same as discussed above for the non-heat transfer case except now the thermal effect of the insulation is accounted for.

Related Topics

Pipes.....	214
Pipe Properties Window.....	214

Common Pipe Input Parameters.....	215
Pipe Model.....	216
Fittings & Losses.....	218
Heat Transfer in Pipes.....	220
Fluid Properties for Pipes.....	238
Optional Input for Pipes.....	238
Add/Remove Pipe Segment.....	241
Reverse Direction.....	242
Pressure Drop in Pipes - Detailed Discussion.....	243

Fluid Properties for Pipes

This folder tab is only visible when variable properties are being modeled. Variable property modeling is enabled in [Analysis Setup](#) on the [Heat Transfer/Variable Fluids panel](#).

For models with variable fluid properties, you can assign different fluid properties to each pipe in the model. This is convenient for modeling systems that are subject to temperature variations and systems that have multiple fluids. The pipe fluid properties are displayed in the [Pipe Properties Window](#), and can be modified by selecting the Fluid Properties button. Besides varying the fluid properties, the viscosity model can be varied from pipe to pipe. Thus some pipes in the model can be modeled as Newtonian, while others as [non-Newtonian](#).

Related Topics

Pipes.....	214
Pipe Properties Window.....	214
Common Pipe Input Parameters.....	215
Pipe Model.....	216
Fittings & Losses.....	218
Heat Transfer in Pipes.....	220
Insulation for Pipes.....	237
Optional Input for Pipes.....	238
Add/Remove Pipe Segment.....	241
Reverse Direction.....	242
Pressure Drop in Pipes - Detailed Discussion.....	243

Optional Input for Pipes

Initial Flow Guess

The [Initial Flow Guess](#) for a pipe can be specified. An appropriate initial guess can decrease the number of iterations required to arrive at a converged solution.

The Solver generates initial guesses on its own, which is usually sufficient to get the Solver going in the right direction so a converged solution results. If this is not the case, user defined guesses can help obtain a solution.

Parallel Pipes

A single [pipe](#) object can be specified as a number of parallel pipes. When multiple pipes are defined, the parameters in the pipe properties window represent only one of the parallel pipes.

Special Condition

The [Special Condition](#) for the pipe can be set to None or Closed. Setting this to closed prevents flow through the pipe, like closing a valve. A symbol (default of "X") will be placed before the pipe label and the pipe will appear dashed, indicating that it is closed.

Design Factors

A design factor can be used to add safety margins into the hydraulic analysis. For pipes, three design factors exist:

- **Pipe Friction** - friction factor is multiplied by the design factor assigned to the pipe.
- **Pipe Fittings & Losses** - total loss factor (K factor or Equivalent Lengths) is multiplied by the design factor assigned to the pipe.
- **Pipe Heat Transfer** - the thermal resistance for each [heat transfer layer](#) is divided by the design factor assigned to the pipe.

Intermediate Pipe Elevations

The inlet and exit elevations of a pipe are equal to the elevations of the upstream and downstream junctions. The pipe elevation is assumed to vary linearly between the junctions. In other words, the pipe is assumed to be perfectly straight.

Modeling a large number of elevation changes with straight pipes would require a large number of junctions, as shown below:

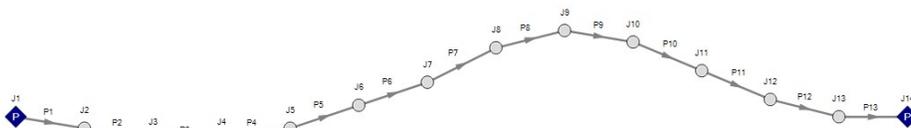


Figure 1: Accounting for elevation changes with many pipes and junctions

This is tedious and not required for overall hydraulic calculations - only the elevation change across a (fluid full) pipe is needed. However, intermediate pressure results along the pipe will be affected.

For example, Figure 1 shows elevation extremes both lower and higher than the elevations at the ends of the pipeline. Low points may exceed maximum allowable pressures, and high points could drop below

vapor pressure. If this is modeled as a single straight pipe these issues would not be easily identified. In fact, if these pipes were replaced with a single pipe it would show no elevation change at all!

Instead of using a large number of pipes, these effects can be accounted for in a single pipe using intermediate pipe elevation feature.

With length defined, check Use Intermediate Pipe Elevations, choose the elevation units, the number of intermediate elevations (a 2 segment pipe has 1 intermediate elevation), and whether the entered values are Point to Point or Length Along Pipe.

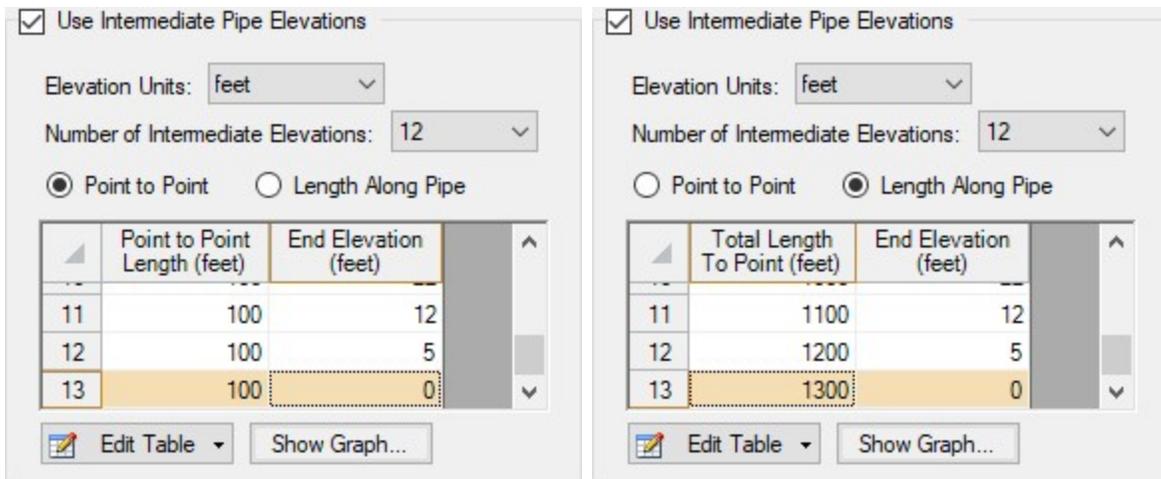


Figure 2: Intermediate Elevations - Identical Profiles defined with both methods

AFT Fathom will ensure the total length matches the length on the Pipe Model tab. It does so by fixing the length of the final section to the difference between the sum of the lengths in the table and the actual length entered by the user. The end elevation is also fixed by the downstream junction. If the user later modifies the downstream junction elevation, the pipe's final elevation section will be automatically updated.

After the intermediate elevations have been entered, a graph showing the data can be displayed. Below is the profile graph of the pipe described above.

Note that a pipe with intermediate elevations entered can be split into multiple pipes connected with branches at the intermediate elevation points. See [Joining and Splitting Pipes](#) for more information.

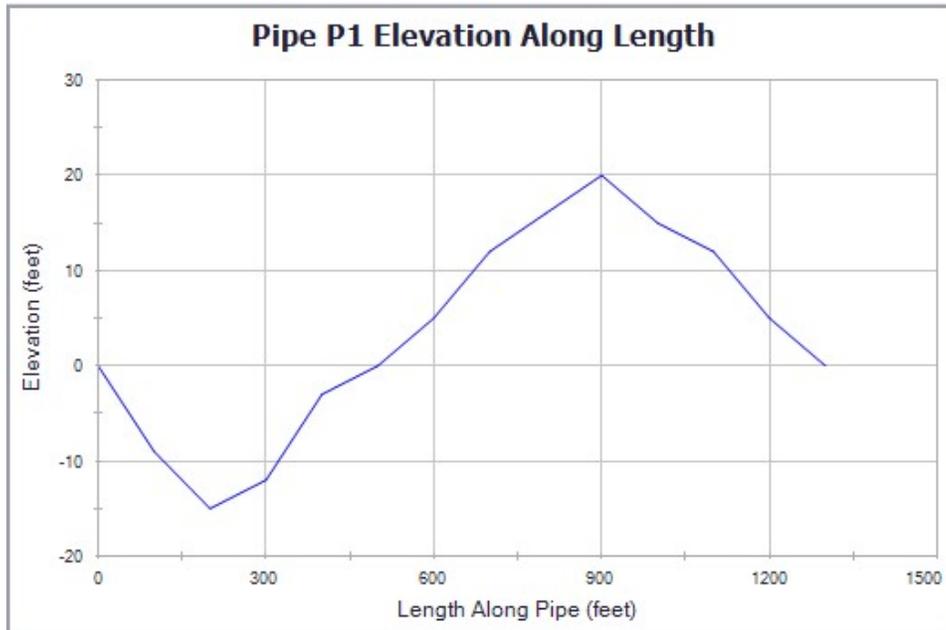


Figure 3: Intermediate Elevation Graph

Related Topics

Special Conditions.....	213
Optional Input for Junctions.....	255
Turn Off Parts of the Model.....	805
Pipes.....	214
Pipe Properties Window.....	214
Common Pipe Input Parameters.....	215
Pipe Model.....	216
Fittings & Losses.....	218
Heat Transfer in Pipes.....	220
Insulation for Pipes.....	237
Fluid Properties for Pipes.....	238
Add/Remove Pipe Segment.....	241
Reverse Direction.....	242
Pressure Drop in Pipes - Detailed Discussion.....	243
Pipe X segment length Y is shorter than the elevation change.....	789
Transfer Results to Initial Guesses.....	800
Make Initial Flow Rate Guesses for Pipes.....	804

Add/Remove Pipe Segment

Visual segments can be added to pipes.

To add or remove segments, use the [Arrange Menu](#), Add/Remove Segments submenu. These options are also available on the Workspace Toolbar.

Note: These segments are useful for creating a clear layout of the system, but have no impact on the solution - a visual bend is not the same as adding a [Bend](#) junction.

Related Topics

Pipes	214
Pipe Properties Window.....	214
Common Pipe Input Parameters.....	215
Pipe Model.....	216
Fittings & Losses.....	218
Heat Transfer in Pipes.....	220
Insulation for Pipes.....	237
Fluid Properties for Pipes.....	238
Optional Input for Pipes.....	238
Reverse Direction.....	242
Pressure Drop in Pipes - Detailed Discussion.....	243

Reverse Direction

Reference Positive Flow Direction

Next to each pipe's ID number is an arrow that indicates the *reference positive flow direction* for the pipe. AFT Fathom assigns a flow direction corresponding to the direction in which the pipe is drawn. You can reverse the reference positive flow direction by choosing Reverse Direction from the [Arrange menu](#) or selecting the reverse direction button on the [Toolbar](#). With exception to some junctions, the reference positive flow direction is used for reference purposes only and does not need to correspond with the real direction of flow calculated by the Solver (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system). If the flow direction obtained by the Solver is opposite of the reference positive flow direction, then the output will show the flow rate as a negative number.

Note: The reference positive flow direction is important for pumps because the pipe directions will define which side is suction vs discharge. Furthermore, reverse flow through components may result in a [Caution message](#) which may or may not be valid depending on the intended direction of flow.

Related Topics

Pipes	214
Pipe Properties Window.....	214

Common Pipe Input Parameters.....	215
Pipe Model.....	216
Fittings & Losses.....	218
Heat Transfer in Pipes.....	220
Insulation for Pipes.....	237
Fluid Properties for Pipes.....	238
Optional Input for Pipes.....	238
Add/Remove Pipe Segment.....	241
Pressure Drop in Pipes - Detailed Discussion.....	243

Pressure Drop in Pipes - Detailed Discussion

Depending on the viscosity model and pipe friction model selected, the calculation of pressure drop in a pipe differs.

Darcy-Weisbach Loss Model

AFT Fathom utilizes the Darcy-Weisbach loss model to relate the Darcy Friction Factor, the pipe geometry, fluid density, and fluid velocity to pressure drop in the pipe. The Darcy Friction Factor differs from the Fanning Friction Factor by a factor of 4.

$$\Delta P = f \frac{L}{D} \frac{1}{2} \rho V^2 \quad (1)$$

This model requires the calculation of a friction factor, as described below.

- Roughness-Based Methods** - This method calculates the friction factor based on the roughness of the pipe wall. Different equations are used based on flow regime - laminar flow uses the standard laminar relationship, while turbulent flow uses the implicit Colebrook-White equation. In the transition range between laminar and turbulent flow, a linearly interpolated value is used. The default transition Reynold's Numbers can be modified in the [Environmental Properties panel](#).

$$f = \frac{64}{Re} \quad \text{Laminar, } Re < 2300 \text{ (default)}$$

$$f = \left[1.14 - 2 \log \left(\frac{\epsilon}{D} + \frac{9.35}{Re \sqrt{f}} \right) \right]^{-2} \quad \text{Turbulent, } Re > 4000 \text{ (default)} \quad (2)$$

- Absolute Roughness (default)** - The absolute average roughness height ϵ is specified directly.
 - Relative Roughness** - The roughness is specified as a ratio ϵ/D .
 - Hydraulically Smooth** - The ratio ϵ/D is set equal to zero.
- Explicit Friction Factor** - The friction factor to be used in the Darcy-Weisbach equation is specified directly.

Frictionless

The pipe will not have any pressure drop across it. This is inherently *unrealistic* behavior, but can be useful for troubleshooting purposes.

Resistance

The pipe resistance relates head loss to volumetric flow rate. In equation form, the head loss is:

$$\Delta H = RQ^2 \quad (3)$$

Using standard relationships, the friction pressure drop is related to volumetric and mass flow rate as follows:

$$\Delta P = \rho g R \left(\frac{\dot{m}}{\rho} \right)^2 = \frac{R g \dot{m}^2}{\rho} \quad (4)$$

Non-Newtonian Pressure Loss

The pressure loss behavior for Non-Newtonian fluids depends on the specific viscosity model selected in Analysis Setup. See [Non-Newtonian fluids](#) for more information.

Helical Tubes

For helical tubes the pressure drop is modified from the equivalent "straight pipe" pressure drop via the following relationship ([Ito 1959](#)):

$$\Delta P_{helical} = \Delta P_{straight} \left[Re \left(\frac{D_{pipe}}{2 r_{coil}} \right)^2 \right]^{0.05} \quad (5)$$

Hazen-Williams Method

AFT Fathom also offers the Hazen-Williams method of specifying irrecoverable loss information. The traditional Hazen-Williams formula ([Zipparro and Hasen editors, 1993](#), page 2.7)

$$V = 1.318 C_{HW} R_h^{0.63} S^{0.54} \quad (6)$$

where

- V = Velocity (ft/s)
- C_{HW} = Hazen-Williams factor
- R_h = Hydraulic Radius (cross-sectional area divided by wetted perimeter)
- S = Hydraulic Gradient ($\Delta H/L$)

can be modified to a more convenient head loss form for AFT Fathom:

$$\Delta H = L \left(\frac{V}{0.55 C_{HW}} \right)^{1.85} D^{-1.17} \quad (7)$$

where

- V = Velocity (ft/s)
- C_{HW} = Hazen-Williams factor
- D = Inner Diameter in feet (four times R_h for circular pipe)
- L = Length of Pipe (feet)

Also see [Hazen Williams NFPA](#)

MIT Equation for Crude Oil

The MIT Equation is appropriate for crude oil and is given by the following equation ([Pipe Line Rules of Thumb Handbook](#)):

$$\frac{dP}{L} = \frac{(0.241 f s Q^2)}{d^5} \quad (8)$$

$$r = \frac{0.0119Q}{dv} \quad \text{MIT Reynold's Number} \quad (9)$$

$$f = \begin{cases} 0.00207 \left(\frac{1}{r} \right) & \text{laminar } 0.1 \leq r \leq 0.135 \\ 0.0018 + 0.00662 \left(\frac{1}{r} \right)^{0.355} & \text{turbulent } r > 0.4 \end{cases} \quad (10)$$

where:

- dP = pressure drop (psi)
- L = length (miles)
- f = friction factor, MIT
- Q = volumetric flow rate (barrels/day)
- s = specific gravity
- d = inside diameter (inches)
- ν = kinematic viscosity (centistokes)

The friction factor calculated by the MIT equations is not the same as the Darcy friction factor. Once the pressure drop (dP) in the pipe is determined, AFT Fathom uses the Darcy equation to back calculate the equivalent Darcy friction factor.

The MIT equation defines the lower bound for the laminar equation as $r=0.1$, but offers no explanation as to the validity of calculations below this point. AFT Fathom continues to use the laminar flow equation for Reynolds numbers below 0.1. The older version of the MIT equations did not have this lower limit, and the results from AFT Fathom 11 agree closely with values from the older equations at these lower Reynolds number values.

For Reynolds numbers in the Indeterminate flow regime, the value for the MIT friction factor is determined by linearly interpolating between the friction factors for the highest laminar Reynolds number and the lowest turbulent Reynolds number.

Miller Turbulent Method

The Miller Turbulent method is appropriate for light hydrocarbons and is given by the following equation ([Pipe Line Rules of Thumb Handbook](#)):

$$Q = \frac{d^{2.5} \left(\frac{dP}{L} \right)}{5.9113 \left(\frac{\rho}{\rho_{water}} \right)^{0.5}} \left[\log \left(\frac{d^3 \left(\frac{\rho}{\rho_{water}} \right) \left(\frac{dP}{L} \right)}{\mu^2} \right) + 4.35 \right] \quad (11)$$

where:

- dP = pressure drop (psi)
- L = length (miles)
- Q = volumetric flow rate (barrels/hour)
- ρ = density (lbm/ft³)
- ρ_{water} = density of water (62.3 lb_m/ft³)
- d = diameter (inches)
- μ = dynamic viscosity (centipoise)

Design Factors

In each pipe you can specify a [Design Factor](#) for the pipe friction. This is a multiplier that is applied to the friction factor calculated with the preceding methods.

Related Topics

Pipes.....	214
Pipe Properties Window.....	214
Common Pipe Input Parameters.....	215
Pipe Model.....	216
Fittings & Losses.....	218
Heat Transfer in Pipes.....	220
Insulation for Pipes.....	237
Fluid Properties for Pipes.....	238
Optional Input for Pipes.....	238
Add/Remove Pipe Segment.....	241
Reverse Direction.....	242

Junctions

This section focuses on the Junctions in the AFT Fathom [Workspace](#) as well as the theories that govern the junctions. Topics included discuss the types of junctions that AFT Fathom provides, the required inputs for each junction, and theory topics that indicate how the inputs to the junction are used.

Contents

General Junction Information	249
Junction Loss Models	261
Area Change	269
Assigned Flow	271
Assigned Pressure	272
Bend	272
Branch	277
Check Valve	277
Control Valve	278
Dead End	281
General Component	282
Heat Exchanger	282
Jet Pump	287
Orifice	287
Pump	291
Relief Valve	330
Reservoir	333

Screen	335
Spray Discharge	337
Tee/Wye	341
Three Way Valve	349
Valve	350
Venturi	352
Volume Balance	353
Weir	355

General Junction Information

The different types of junctions on the Toolbox offer a great degree of freedom in assembling a pipe network model. While there is some redundancy in capabilities between the junctions, each of the junctions offers specialized features to allow you to prepare a good conceptual model of the physical system of interest. Table 1 compares the features of the junction types.

There is a junction properties window for each junction type. Each junction Properties window falls into one of [two basic window formats](#), depending on the number of connecting pipes allowed for that junction type.

Note: AFT Fathom does not prevent negative flow through junctions. Should this occur, AFT Fathom will use the same loss factor or pressure drop data referenced to the defined upstream pipe, just as in forward flow. Because it is possible that the loss factor or pressure drop data would not be the same for both flow directions, the flow solution could be misleading. For this reason, it is important that you either properly define the pipe flow directions for junctions or pay close attention to the flow directions that result. Should negative flow occur in any junction, AFT Fathom will give a caution in the output.

Junctions

Table 1: Comparison of junction capabilities

Junction Type	Connecting Pipes	Special Condi-tions	Restricted Area	Custom Library	Window Format
Area Change	2			Yes	1
Assigned Flow	1	Yes		Yes	1
Assigned Pres-sure*	1/1-25			Yes	1
Bend/Elbow	2		Yes	Yes	1
Branch	2-25	Yes		Yes	2
Check Valve	2	Yes	Yes	Yes	1
Control Valve	2	Yes		Yes	1
Dead End	1				1
General Com-ponent	2	Yes		Yes	1
Heat Exchanger	2	Yes		Yes	1
Jet Pump	2	Yes		Yes	1
Orifice	2		Yes	Yes	1
Pump	2	Yes		Yes	1
Relief Valve	1-2	Yes	Yes	Yes	1
Reservoir	1-25			Yes	2
Screen	2		Yes	Yes	1
Spray Dis-charge	1-4	Yes	Yes	Yes	2
Tee/Wye**	3				2
Three-Way valve**	3	Yes		Yes	2
Valve	1-2	Yes	Yes	Yes	1
Venturi	2		Yes	Yes	1
Volume Bal-ance	2				1

Junctions

Weir	2	Yes	Yes	Yes	1
------	---	-----	-----	-----	---

*Assigned Pressure junction connects to only one pipe for static pressure, and up to 25 for stagnation

**Tee/Wye and Three-way valve junctions have a unique format closer to Format #2 than Format #1

Related Topics

Two Window Formats.....	251
Common Junction Input Parameters.....	254
Optional Input for Junctions.....	255
Specifying Losses.....	257
Conventions for Specifying Basis Area.....	257
Reference Density.....	258
Equal Percentage Valves.....	260

Two Window Formats

Format #1: Junctions with One or Two Connecting Pipes

The first basic window format is for junction types that allow only one or two connecting pipes (see Figure 1). While junctions do not have an explicitly defined flow direction like pipes do, typically those with two pipes or less adopt a direction from the connecting pipes. For these junction types, upstream and downstream pipes are recognized by AFT Fathom based on the [reference positive flow direction](#) of the connecting pipes. The upstream and downstream pipes are displayed separately near the top of the window.

In this first basic window type, it is generally important to have the pipe's reference positive flow directions specified in the physically correct directions. A good example of this is a [pump junction](#). A pump will add pressure to the system, but it does so only in the direction of positive flow through the connecting pipes. The Pump junction interprets where to add the pressure based on the directions of the connecting pipes.

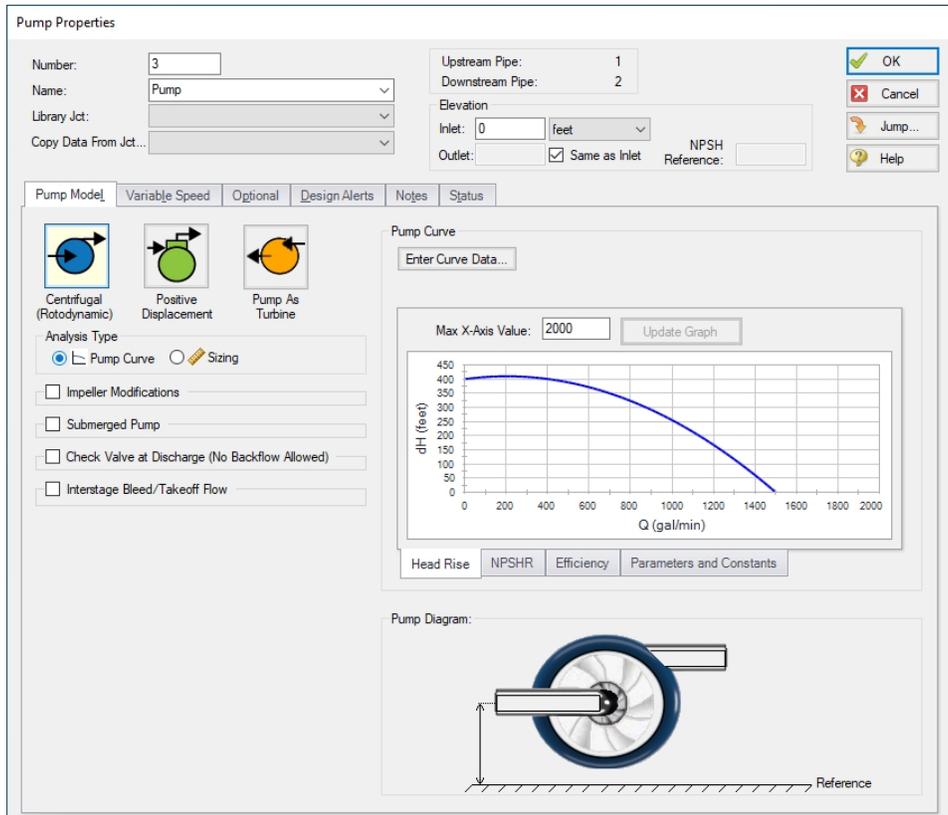


Figure 1: Properties window format #1 (fixed connecting pipe display)

Format #2: Junctions with More Than Two Connecting Pipes

The second basic window format for junction Properties Windows is for junctions that allow more than two connecting pipes (see Figure 2). These junctions typically allow up to twenty-five connecting pipes; the exceptions are the [Three-Way Valve](#) and [Tee/Wye](#) junctions, which allow only three.

Because the number of pipes connected to a junction may vary, the second basic format uses a table sized according to the number of connecting pipes.

An example of this second window format is the [Branch Properties](#) window, shown in Figure 2, which has three connecting pipes. To determine AFT Fathom's interpretation of the model connectivity for this second window format, you can review the contents of the table areas.

K factors in the first column will be applied for flow into the pipe and out of the junction, while K factors in the second column will be applied for flow out of the pipe and into the junction.

Junctions

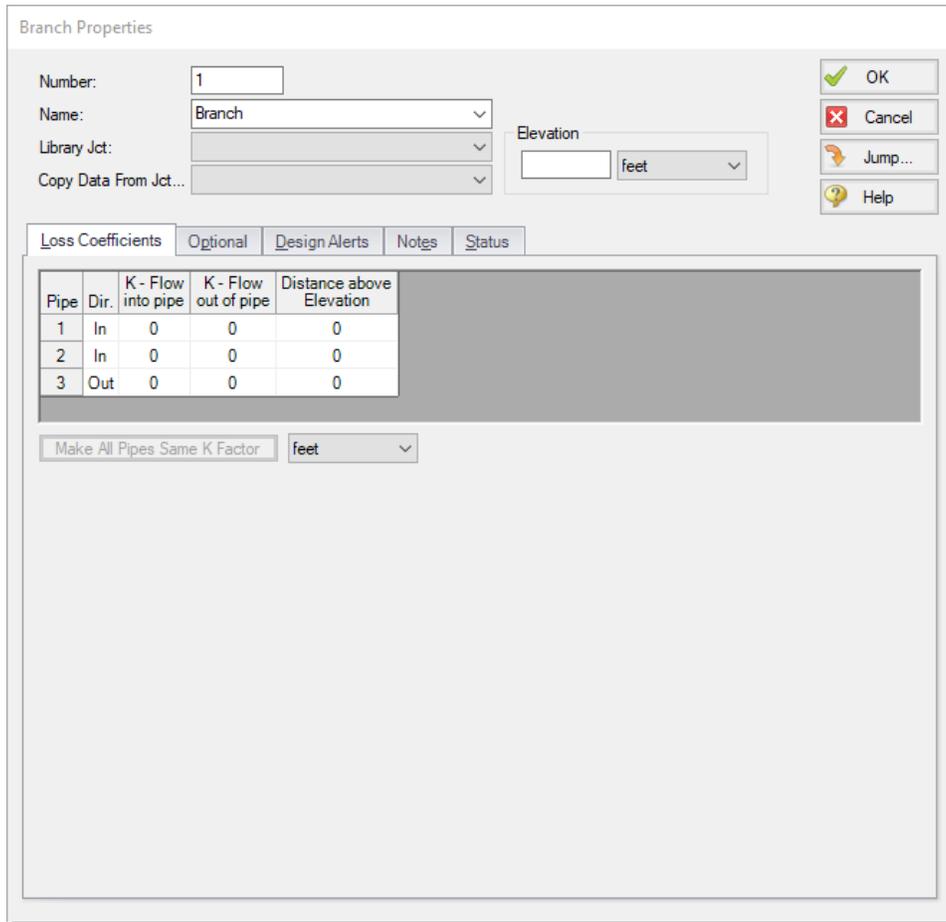


Figure 2: Properties window format #2 (pipes displayed in a table)

Related Topics

General Junction Information.....	249
Common Junction Input Parameters.....	254
Optional Input for Junctions.....	255
Specifying Losses.....	257
Conventions for Specifying Basis Area.....	257
Reference Density.....	258
Equal Percentage Valves.....	260

Common Junction Input Parameters

Number

Every junction must have a unique ID number. When new junctions are created, a junction number is automatically allocated.

The junction number can be changed by the user to any unique integer value greater than zero and less than 100,000. Junction numbers are only identifiers and have no effect on the model behavior.

Name

A name can be assigned to each junction for reference purposes. Names do not need to be unique. The name can be shown on the Workspace and in the Output window.

Library List

A list of all available [custom components](#). It displays components from any [connected library](#).

Selecting a component from this list automatically retrieves the junction settings and populates the properties window. Changing the input manually will deselect the component in the list.

Copy Data From Junction

The Copy Data From Junction dropdown list allows you to copy all or some of an existing junctions' properties to the current junction. Selecting an existing junction will prompt the user on what parameters should be copied over.

In child [scenarios](#), Copy Data From Junction lists "Parent Junction Data," allowing the user to copy all or some parameters from the parent junction.

Pipe Connectivity

The connected pipes are displayed. This is helpful to determine potential [connection](#) issues and easily identify which pipes are upstream and downstream of the junction.

Elevation

Every junction must have an elevation reference. This value is the centerline elevation of the connected pipe(s).

If multiple pipes are connected to a single junction, there are multiple elevations that must be defined. By default, all connected pipe centerlines are assumed to be at the same elevation. A different inlet/outlet elevation can be specified by unchecking the "Same as Inlet" box for junctions with only two connecting pipes.

For junctions with more than two pipe connections, these elevations can be specified in the Pipe Depth or Loss Coefficients tab, depending on junction.

Elevations are used to account for pressure changes due to gravity. There can be a pressure changes due to elevation change in a pipe or a junction. Junction elevation changes will have no effect on the flow solution, but will offset the local static pressure by the hydrostatic pressure difference.

A default elevation can be specified in Parameter Options, automatically setting new junctions to the specified elevation.

Jump Feature

Quickly move to the next junction. See [Jump Feature](#).

Related Topics

General Junction Information.....	249
Two Window Formats.....	251
Optional Input for Junctions.....	255
Specifying Losses.....	257
Conventions for Specifying Basis Area.....	257
Reference Density.....	258
Equal Percentage Valves.....	260
Pipe X length is shorter than the elevation change.....	789

Optional Input for Junctions

Initial Guesses

A user defined initial guess can be entered for pressure (or head) and temperature (if heat transfer is modeled) for most junctions. This is only an initial guess for the Solver - the solution *will* vary from this value, and the value has **no effect** on the end solution.

A good initial guess can be helpful to decrease the number of Solver (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system) iterations required, and potentially help with convergence issues.

Special Conditions

The [Special Condition](#) can be changed for some junctions. For example, a valve can be set to closed, preventing all flow. A symbol (default of "X") will be placed before the junction label indicating that a special condition is applied.

Design Factor

A design factor can be used to add safety margins into the hydraulic analysis. For junctions, three types of design factors exist:

- **Pump Junctions** - head rise at the [pump](#) is multiplied by the design factor assigned to the pump.
- **Heat Exchanger Junctions** - total rate of heat transfer is multiplied by the Heat Transfer design factor assigned to the [heat exchanger](#). The total loss factor is multiplied by the Friction Loss design factor assigned to the heat exchanger.
- **All other junctions** - total loss factor is multiplied by the design factor assigned to the junction.

Note: The design factor for pumps has the opposite effect on a system that the design factor for other junction types has. A design factor of 1.1 for junctions with a pressure loss will give an extra 10% pressure drop at that junction, adding resistance to the system. A design factor of 1.1 for the pump junction will give an extra 10% pressure rise, effectively modeling the pump as having extra capacity than expected. To add a 10% safety margin to the pump junction, a design factor of 0.9 should be used.

Workspace Icon

The junction icon can be changed to better represent a specific junction type. This is accomplished by clicking the Change Icon button. Custom images can be added by clicking the Add Image button. The size of the junction can be changed via [Workspace Layers](#).

Restricted Area

This is an optional parameter in several junction Properties windows which describes the effective flow area through the junction. This is done for the purpose of assessing cavitation. In most cases, information on the size of the restricted area must be obtained from handbook or test data.

Related Topics

Special Conditions	213
Optional Input for Pipes	238
Turn Off Parts of the Model	805
General Junction Information	249

Two Window Formats	251
Common Junction Input Parameters	254
Specifying Losses	257
Conventions for Specifying Basis Area	257
Reference Density	258
Equal Percentage Valves	260
Transfer Results to Initial Guesses	800
Make Initial Flow Rate Guesses for Pipes	804
Cavitation May Exist at Junction X	776

Specifying Losses

Local losses can be specified in two ways. The most flexible way of including a loss factor in an AFT Fathom model is to associate it with a [junction](#) specifically defined for that type of loss. This allows you to define a loss as a function of a flow parameter such as Reynolds number or volumetric flow rate, or as a function of the area change ratio between the upstream and downstream pipes. This method results in the greatest clarity.

Secondly, you can include loss factors with pipes. When you specify [Fittings & Losses](#) in the [Pipe Properties Window](#), the loss is assumed to be distributed evenly along the pipe length, much like friction losses. For this reason a pipe-associated loss factor is referred to as a distributed loss. The losses at junctions, on the other hand, are point losses.

Related Topics

General Junction Information	249
Two Window Formats	251
Common Junction Input Parameters	254
Optional Input for Junctions	255
Conventions for Specifying Basis Area	257
Reference Density	258
Equal Percentage Valves	260

Conventions for Specifying Basis Area

[Loss factors](#) are area dependent. When specifying a loss factor at a junction, the base area is of critical importance. By default, AFT Fathom adheres to the following convention: whenever it makes sense physically, the basis area is always the upstream pipe flow area. However, this can be modified by the user. In most junction Properties windows (on the Loss Model tab), the basis area can be specified as the upstream pipe, downstream pipe, user specified area, or user specified diameter.

However, some junction types, especially those that allow three or more connecting pipes, specify loss factors based on each connecting pipe flow area, whether upstream or downstream. An example of this is a [Wye](#) in which each of the connecting pipes has a different diameter. The idea of specifying a loss factor based on upstream flow area loses meaning because there may be more than one upstream pipe. In these cases the loss factor is referenced to the base area of the actual downstream pipe(s) where it connects to the junction. The types of junctions that do not always follow the upstream pipe area con-

vention are the [Tee/Wye](#), [Branch](#), [Reservoir](#), [Assigned Pressure](#), and [Assigned Flow](#). Other exceptions are [Valves](#) and [Relief Valves](#) specified as exit valves, and [Orifices](#) specified as an exit.

Related Topics

General Junction Information.....	249
Two Window Formats.....	251
Common Junction Input Parameters.....	254
Optional Input for Junctions.....	255
Specifying Losses.....	257
Reference Density.....	258
Equal Percentage Valves.....	260

Reference Density

A relationship can be drawn for many components between *flow energy* and *flow rate*. This can be a decrease (e.g., a loss across a valve) or increase in energy (e.g. energy added by a pump). The specific parameter used to represent *flow energy* can vary depending on application. It can be given in terms of *pressure change* or *head change*, and there is a critical difference between the two.

Consider a centrifugal turbomachine (a pump or radial compressor): Kinetic energy is imparted to the fluid by accelerating it along the radius of the impeller. If the rate of rotation is held constant, a denser fluid will have more energy imparted to it than a lighter weight fluid. This would give rise to higher pressures. In other words, the *pressure change* depends on the density of the fluid.

Pump curves are generally provided in terms of head. This is convenient because head is a unit of specific energy - or energy per weight. Therefore, it does not vary with fluid density like pressure does. The head added to a fluid by a pump at a fixed speed, for example, will be the same for water and liquid mercury.

However, head relationships are not always available or meaningful. **If a pressure vs flow relationship is used, a reference density must be specified.** This is the fluid density used during testing and creation of the curve by the component manufacturer. By default, AFT software will use the density of the working fluid in the model defined on the [Fluid panel](#), but this can be changed using the Reference Density setting on certain junctions.

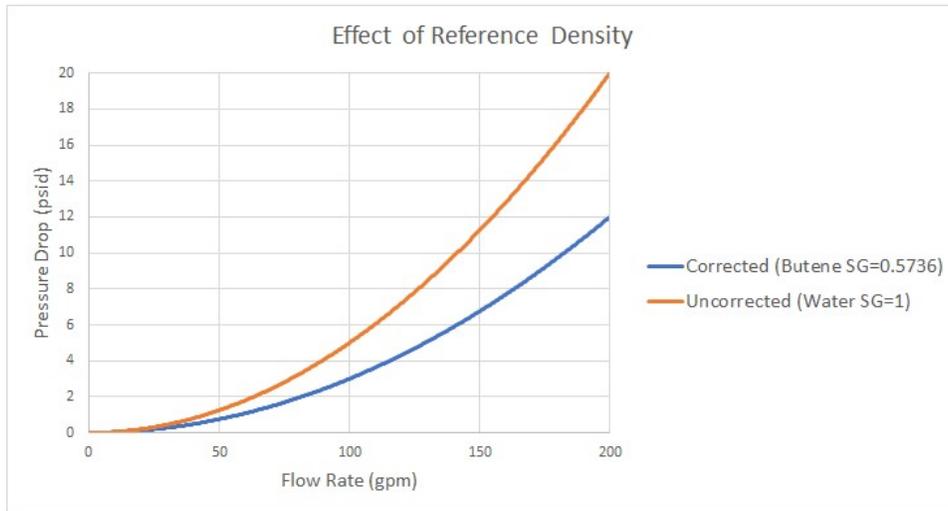


Figure 1: Visual comparison of Pressure Loss Curves for differing densities (head curves would be identical for this component)

Affected Parameters

There are several parameters that are affected by the density of the fluid:

- **Pressure** - As discussed above
- **Mass Flow** - The mass term contains information about the density of the fluid
- **Power** - More power is demanded by a pump to meet the head curve for a denser fluid. While head is independent of density and therefore often reported by manufacturers, power is not independent of density. Therefore, a proper specification sheet that includes power must also include the reference fluid density.

Accounting for Reference Density

[Pumps](#) and [Resistance Curves](#) allow for the specification of a reference density if one of the above parameters is selected. The selection appears below the curve fit area as shown in Figure 2.

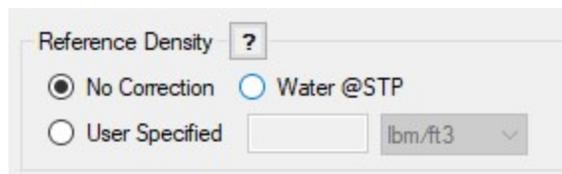


Figure 2: Reference Density Selection

Reference Density - density of the fluid that the manufacturer used during testing to generate the pressure loss vs flow data for the component.

System Fluid Density - density of the fluid defined within the AFT model on the [Fluid panel](#).

If the System Fluid Density significantly differs from the Reference Density, then a correction should be made using the Reference Density option.

- **No Correction (default)** - This selection assumes that the component's pressure loss data was developed by the manufacturer using the same fluid that is defined in the model on the [Fluid panel](#).
- **Water @STP** - The reference density is that of water at standard temperature and pressure. If the Analysis Setup Fluid differs, then the entered curve will be corrected.
- **User Specified** - This directly specifies the reference density. The user should enter the density of the fluid that the manufacturer used during testing to generate the pressure loss vs flow data. If the Analysis Setup Fluid differs, then the entered curve will be corrected.

Related Topics

General Junction Information.....	249
Two Window Formats.....	251
Common Junction Input Parameters.....	254
Optional Input for Junctions.....	255
Specifying Losses.....	257
Conventions for Specifying Basis Area.....	257
Equal Percentage Valves.....	260

Equal Percentage Valves

Equal Percentage valves have a valve plug shaped so that each increment in valve percent open (lift) increases the flowrate by a certain percentage of the previous flow.

The relationship between valve lift and orifice size is not linear but logarithmic.

It is expressed mathematically as follows:

$$C_v = \frac{e^x}{\tau} C_{v,max}$$

$$x = \ln(\tau)H$$

where:

- τ = valve rangeability (ratio of the maximum to minimum controllable flowrate)
- H = valve percent open or lift (0 = closed; 1 = fully open)
- $C_{v,max}$ = full open valve loss value

Related Topics

General Junction Information	249
Two Window Formats	251
Common Junction Input Parameters	254
Optional Input for Junctions	255
Specifying Losses	257
Conventions for Specifying Basis Area	257
Reference Density	258

Junction Loss Models

There are a number of loss models available in AFT Fathom to account for frictional losses at the junctions in the system. See the below topics for more information on how losses are calculated for each of the available models.

K Factor Loss Model

AFT Fathom models K-factor component losses according to the following equation:

$$\Delta P_{loss} = K \frac{1}{2} \rho V^2$$

where K is commonly referred to as the loss factor.

Table 1 lists the sources for the loss models used in AFT Fathom. The losses implemented directly in the code were chosen on the basis of ease of use. Many loss factor types are functions of the flow, and thus too general to be easily incorporated (see the [Resistance Curve](#) topic for flow dependent loss applications).

Table 1: Loss model references

Junction Type	References
Bend	Crane 1988, A-29 Miller 1990, 207-209
Area Change	Crane 1988, A-26 Idelchik 2007, 246 & 254
Orifice	Idelchik 2007, 258-259 & 262
Screen	Idelchik 2007, 598
Tee/Wye	Idelchik 2007, Chapter 7 Miller 1990, 315-316 & 323
Valve	Ingersoll-Rand 1970 Crane 1988, A-29

[Crane \(1988\)](#) offers good general purpose correlations for modeling irrecoverable losses in pipe systems.

Another lesser known source of loss factor information is [Idelchik's Handbook of Hydraulic Resistance \(2007\)](#). This reference is indispensable for the engineer who must make detailed hydraulic assessments of pipe systems in which so-called minor losses play a significant role. The reference is voluminous in scope and full of tables, charts and equations for calculating loss factors for almost any pipe arrangement.

[Miller's Internal Flow Systems \(1990\)](#) is another reference offering good general purpose hydraulic data.

Related Topics

Flow is negative through Junction X. Loss factors may be in error.....	785
Junction Loss Models.....	261
Kv and Cv Loss Model.....	263
Resistance Curve Loss Model.....	264
3-K (Darby) Method.....	266
Equivalent Length.....	267
Bend Loss Model.....	274
Area Change Loss Model.....	269
Orifice Loss Model.....	288
Screen Loss Model.....	336
Tee/Wye Loss Model.....	342
Valve Modeling Closures with K Factors.....	352

Kv and Cv Loss Model

AFT Fathom allows the user to enter valve coefficient data in terms of K_v or C_v . Conversions are handled seamlessly in the background. K_v is the metric equivalent to C_v .

Manufacturer Testing in Standard Units

C_v (Coefficient of flow for Valves) is a single parameter which relates fluid density (given in units of specific gravity), pressure drop (in psi), and flow rate (in gallons per minute).

$$C_v = Q \sqrt{\frac{SG}{\Delta P}}$$

One can get a better understanding of the role of C_v by setting the pressure drop (dP) and SG to 1, in which case C_v is equal to flow in gallons per minute. C_v could be considered the flow capacity (in gpm) of a given fluid (i.e. a given SG) for a given pressure drop. Or, one may wish to calculate the pressure drop through a valve, with a given C_v at a given flow rate.

$$\Delta P = SG \left(\frac{Q}{C_v} \right)^2$$

Manufacturer Testing in Metric Units

If the engineer is provided with valve data in metric units, they are also likely to be provided a value for K_v and not C_v .

$$K_v = Q \sqrt{\frac{SG}{\Delta P}}$$

$$K_v = 0.8646 C_v$$

$$C_v = 1.157 K_v$$

The only difference is the units used for Q and dP . K_v uses flow units of m^3/hr and pressure drop units of bar.

If one wishes to calculate pressure drop through a valve, with a given K_v , at a given flow rate, the same relationship applies.

$$\Delta P = SG \left(\frac{Q}{K_v} \right)^2$$

Related Topics

Junction Loss Models.....	261
K Factor Loss Model.....	261
Resistance Curve Loss Model.....	264
3-K (Darby) Method.....	266
Equivalent Length.....	267

Resistance Curve Loss Model

The resistance curve loss model is available in many different junctions. It is useful when measured flow vs pressure loss data is available for a component.

Clicking Enter Curve Data will open a new window where the input can be entered into a table. Several options for the Flow Parameter are available using the dropdown in the column header, and the appropriate units can be selected. Pressure loss can be alternatively entered as a head value.

Fill As Quadratic (Using a Single Known Data Point)

It is not uncommon to obtain only a single data point from an equipment manufacturer. For example, the data might be 3 psid pressure drop at 1 lbm/sec. In this case, the known data point can be entered into the first row of the data table. Then, selecting Fill As Quadratic will automatically insert another two data points to be used with a quadratic curve fit. The first data point will be zero pressure loss at zero flow which is physically realistic for a static component. Note that elevation differences within a junction are accounted for externally by the Solver. The last data point will be a quadratic extrapolation using the zero point and the originally provided data point. Clicking Generate Curve Fit Now will create the curve fit constants and complete the resistance curve definition.

This method is often a reasonable approach to define a static component because the pressure drop characteristics of most equipment follows the square law. As the flow rate increases, the pressure drop increases with the square of the flow rate. Defining the component with a curve fit provides a robust solution where the junction can still provide a reasonable pressure loss even if the flow rate in the model is changed. This also aids in numerical convergence when running a transient simulation with the [XTS Module](#).

In our example, an increase in the flow rate to 2 lbm/sec would result in a pressure drop of 12 psid. Although this may not be exactly correct depending on the component, it is much better than assuming a flat 3 psid drop for all flow rates, which we know for certain is not correct.

Table 1: Example data points for pressure drop modeling using manufacturer data

Data Point	Flow Rate (lbm/sec)	Pressure Drop (psid)	Comment
1	0	0	Always valid for passive equipment
2	1	3	Given by manufacturer
3	2	12	Extrapolated assuming square law

Curve Fitting

[Curve fitting](#) can be performed via a polynomial or interpolated data. The Curve Fit Order can also be changed to achieve a better fit to the data, and the curve fit constants are displayed in a table.

Note: Unrealistic behavior could occur if the curve fit does not pass through the origin of the graph. This may negatively impact convergence when running transient simulations that have zero flow through the component at a point in time. See the [Troubleshooting Resistance Curves](#) topic for more information.

Reference Density

Options to correct for a reference density are available when using mass or pressure parameters. This is useful for when the pressure loss data was measured with a different working fluid than what is used in the model. See the [Reference Density](#) topic for more information.

Related Topics

Junction Loss Models.....	261
K Factor Loss Model.....	261
Kv and Cv Loss Model.....	263
3-K (Darby) Method.....	266
Equivalent Length.....	267
Troubleshooting Resistance Curves.....	801
General Component.....	282
Heat Exchanger.....	282

Related Examples

Heat Exchanger System	866
Cooling System	895

3-K (Darby) Method

The 3-K method described in [Darby, 2001](#), pp. 209-211, is used for high accuracy of predicted loss factors at low Reynolds numbers. The 3-K loss model is available in the [valve](#) and [bend](#) junctions where there is a dropdown menu to select from standard types of elbows and valves. After selecting an option in this menu, the 3 K's will be displayed at the bottom of the properties window. These values come from the chart in Darby on pages 210-211. The user can also select User Specified in the 3-K (Darby) dropdown menu to input their own values for K_1 , K_i , and K_d .

$$K_f = \frac{K_1}{Re} + K_i \left(1 + \frac{K_d}{D_{n,in}^{0.3}} \right)$$

Where:

- K_f = Resistance coefficient
- K_1 = 3-K method constant
- K_i = 3-K method constant
- K_d = 3-K method constant
- Re = Reynolds Number
- $D_{n,in}$ = Inner diameter

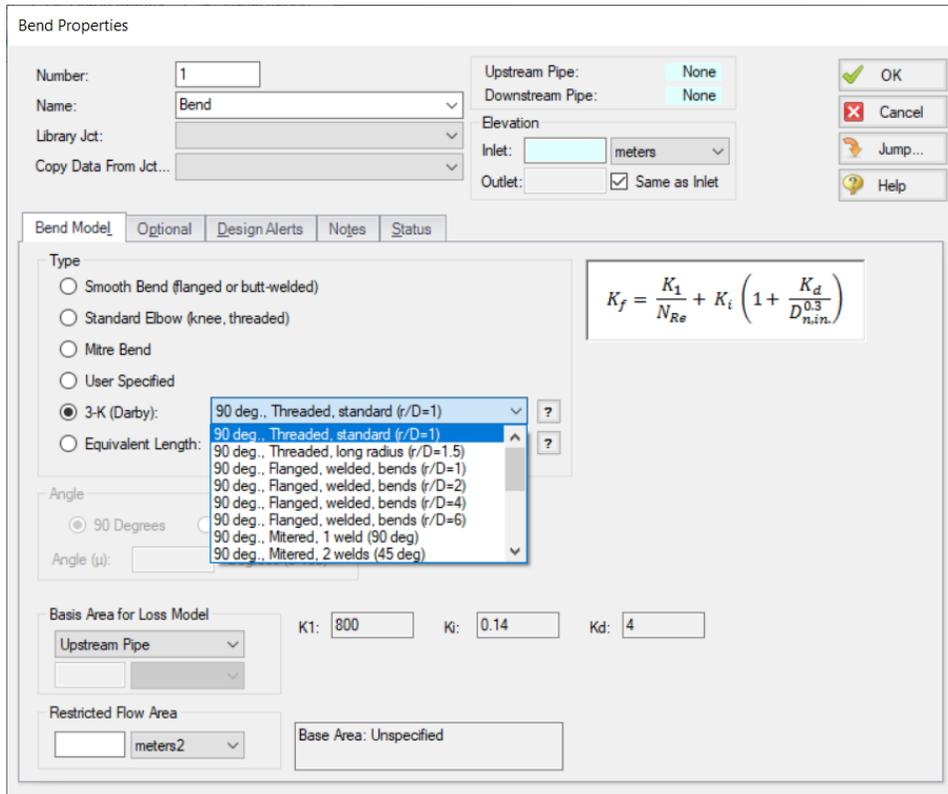


Figure 1: Bend Properties window when the 3-K (Darby) method is turned on

Related Topics

Laminar and Non-Newtonian Corrections Panel.....	123
Adjusted Turbulent K Factor Method (ATKF).....	125
Improved ATKF Method.....	127
Note about Laminar Corrections for Fittings.....	130
Junction Loss Models.....	261
K Factor Loss Model.....	261
Kv and Cv Loss Model.....	263
Resistance Curve Loss Model.....	264
Equivalent Length.....	267

Equivalent Length

The use of equivalent lengths is supported by AFT Fathom in the [Valve junction](#), [Bend junction](#), and the pipe [Fittings & Losses](#). The options for bends and valves come from table 23.4.3.1.1 in [NFPA 13](#). If you are not using the NFPA 13 standard then using a user-specified equivalent length is recommended. To enable NFPA reporting for use with equivalent length losses see the [NFPA panel](#) topic.

The loss factors in table 23.4.3.1.1 assume that a Hazen-Williams C-Factor of 120 and schedule 40 steel piping are being used. If your reference pipe is defined using a different Hazen-Williams factor or different pipe schedule, then the equivalent length will be adjusted by equation 6.2 (Equation 1 below) on page 103 of [Brock, Pat D. 2012](#).

$$L_{eq} = L_1 \left(\frac{D_{eq}}{D_1} \right)^{4.87} \left(\frac{C_{eq}}{C_1} \right)^{1.85} \quad (1)$$

Where:

- L_{eq} = new equivalent length
- L_1 = original length
- D_{eq} = diameter of new equivalent length
- D_1 = original diameter
- C_{eq} = C-Factor for new equivalent length
- C_1 = original C-Factor

This equation is intended for taking an equivalent length and adjusting for a new equivalent length based off changes to diameter and C-Factor. This is a general equation, but NFPA table data from table 23.4.3.1.1 is only for schedule 40 steel and a C-Factor of 120. Therefore, the adjusted equivalent length can be written as shown in Equation 2.

$$L_{eq} = L_1 \left(\frac{D_{eq}}{D_{40}} \right)^{4.87} \left(\frac{C_{eq}}{120} \right)^{1.85} \quad (2)$$

Equation 2 (adjusted from Brock et. al. Equation 6.2) will predict an adjusted equivalent length that coincides with the NFPA 13 equivalent length modifier 23.4.3.1.3.1 and the C-Value Multiplier from table 23.4.3.2.1. Note that if the reference piping is using a friction model other than Hazen-Williams, C_{eq} is assumed to be 120 and the equivalent length will not be adjusted due to friction.

Related Topics

NFPA Panel.....	175
Hazen-Williams NFPA.....	177
Fittings & Losses.....	218
Fittings & Losses Library.....	717
Junction Loss Models.....	261
K Factor Loss Model.....	261
Kv and Cv Loss Model.....	263
Resistance Curve Loss Model.....	264
3-K (Darby) Method.....	266

Area Change

The Area Change junction type must always have two connecting pipes. This junction type allows you to model the irrecoverable loss that occurs at the area transition between two pipes, whether expansion or contraction.

The Area Change Properties Window follows the first of the two basic [Properties Window formats](#), displaying the connecting pipes in a fixed format. A flow direction through the junction is adopted from the defined directions of the two connecting pipes.

The Area Change Properties Window offers two standard types of area change losses: the Conical Transition and the Abrupt Transition.

The critical parameter that influences the magnitude of the area change loss is the area ratio between upstream and downstream pipes. Until you have input the information for the upstream and downstream pipes that allow for flow area determination, the standard Area Change junction loss factor is not calculated. For conical transition geometries, you also need to specify the conical angle. This angle has a range of 0-180 degrees, with a 0 degree angle representing a transition taking place over an infinitely long length, and a 180 degree angle representing an abrupt transition. A depiction of the angle is shown in Figure 1.

If the standard loss factors provided are inadequate for your application, you may specify your own custom loss factor referenced to the basis area you specify.

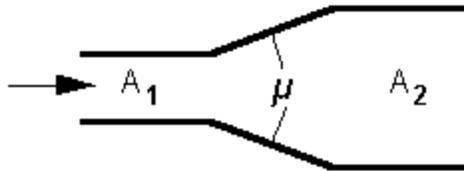


Figure 1: Angle for an area change

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Area Change Loss Model.....	269

Area Change Loss Model

Two standard area change loss geometries are available: the conical transition and the abrupt transition.

The conical expansion correlation ([Crane 1988](#), A-26) is

$$K_{up} = 2.6 \left(\sin \frac{\mu}{2} \right) \left(1 - \frac{A_{up}}{A_{down}} \right)^2 \quad \mu < 45^\circ$$

$$K_{up} = \left(1 - \frac{A_{up}}{A_{down}} \right)^2 \quad \mu > 45^\circ$$

The conical contraction correlation (Crane 1988, A-26) is

$$K_{up} = \frac{0.8 \left(\sin \frac{\mu}{2} \right) \left(1 - \frac{A_{down}}{A_{up}} \right)}{\left(\frac{A_{down}}{A_{up}} \right)^2} \quad \mu < 45^\circ$$

$$K_{up} = \frac{0.5 \left(1 - \frac{A_{down}}{A_{up}} \right) \sqrt{\sin \frac{\mu}{2}}}{\left(\frac{A_{down}}{A_{up}} \right)^2} \quad \mu > 45^\circ$$

The abrupt expansion correlation (Idelchik 2007, 246) is

$$K_{up} = \left(1 - \frac{A_{up}}{A_{down}} \right)^2$$

The abrupt contraction correlation (Idelchik 2007, 254) is

$$K_{up} = \frac{0.5 \left(1 - \frac{A_{down}}{A_{up}} \right)^{0.75}}{\left(\frac{A_{down}}{A_{up}} \right)^2}$$

Note: While a 180 degree conical transition is the same as an abrupt transition, Crane's conical contraction equation gives a different result at 180 degrees compared to Idelchik's abrupt contraction equation. Therefore, it is up to the user to determine which one to use.

All K-factor equations are provided here in terms of the upstream pipe. The K-factor in terms of the downstream pipe can be found as follows:

$$K_{down} = K_{up} \left(\frac{A_{down}}{A_{up}} \right)^2$$

Related Topics

K Factor Loss Model.....	261
Bend Loss Model.....	274
Orifice Loss Model.....	288
Screen Loss Model.....	336
Tee/Wye Loss Model.....	342
Valve Modeling Closures with K Factors.....	352
Area Change.....	269

Assigned Flow

The Assigned Flow junction type allows you to connect one pipe and to input a known flow rate entering or leaving at a particular location. An irrecoverable loss can also be included.

The Assigned Flow Properties window follows the first of the [two basic Properties Window formats](#), displaying the connecting pipe information in a fixed format. Depending on whether you specify the junction as an inflow or outflow type, the required connecting pipe (upstream/inlet or downstream/outlet) will be enabled while the other pipe is disabled. Note that if the flow is specified as Inflow (to the system), then the reference positive flow direction of the connected pipe must be away from the junction (i.e. the junction is upstream of the pipe).

The Assigned Flow junction will automatically switch the inflow/outflow parameter based on the reference flow direction of the connected pipe.

The Assigned Flow junction type allows you to specify positive flow rates as volumetric or mass flow rates. Negative flow rates are not accepted. If, for example, you have an outflow type junction and your system is physically flowing in, you cannot assign a negative flow rate to the junction. You must reverse the connecting pipe flow direction to be consistent with the actual direction and change from an outflow type to an inflow type.

Transient

([XTS Module](#) Only) On the Transient tab, a flowrate transient can be entered. For more information on transient data, including event transients, see [Junction Transient Data](#).

Special Conditions

You can set a [Special Condition](#) for an Assigned Flow junction, which will turn the flow off and make it act like a Dead End.

Loss Factor

On the Optional tab, a loss factor can be entered to account for losses due to fittings or components at the entry/exit to the connected pipe.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261

Assigned Pressure

The Assigned Pressure junction allows for the specification of a fixed boundary pressure and temperature (if heat transfer is enabled).

This junction is very similar to the [Reservoir](#) in that it fixes a pressure at a boundary. However, the Assigned Pressure allows the definition of either a [static or stagnation pressure boundary](#), whereas Reservoirs are always stagnation pressure. Typically, any pressure gauge on a pipe will be reporting static pressure - it is important to select static pressure if this is the actual boundary in the real system. A Reservoir would be inappropriate to represent such a piping connection with non-zero flow.

If stagnation is selected, the Assigned Pressure accepts the connection of up to twenty-five pipes. Static selection limits the connection to one pipe.

When heat transfer is modeled, a temperature must also be specified. There is also the option to Balance Energy in the junction - this adjusts the temperature in order to make incoming and outgoing energy balance. This option is only appropriate for closed loop systems, read more here: [Open vs Closed Systems - Balancing Energy](#).

Loss Factor

On the Loss Coefficients tab, loss factors can be entered to account for losses due to fittings or components at the entry/exit to each of the connected pipes.

Transient

([XTS Module Only](#)) On the Transient tab, a pressure transient can be entered. For more information on transient data, including event transients, see [Junction Transient Data](#).

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261

Bend

The Bend junction type must always have two connecting pipes. This junction type allows you to model the irrecoverable loss that occurs when the flow direction changes through a bend or elbow. The Bend Properties window follows the first of the two basic [Properties Window formats](#), displaying the connecting pipes in a fixed format. A flow direction through the junction is adopted from the defined directions of the two connecting pipes.

Standard Loss Types

The Bend Properties window offers several standard types of bend losses: the Smooth Bend, the Standard Elbow (the standard short radius threaded elbow), the Mitre Bend, and the User Specified K-factor. The window shows a schematic of the selected geometry for these three standard loss types. Until you have input the information for the upstream pipe that allows flow area determination, the standard Bend junction loss factor is not calculated. For smooth radius bends, you also need to specify r/D , which is the radius of curvature divided by the pipe diameter. At r/D values less than one, the Reynolds number must be known to calculate the loss factor, and therefore cannot be displayed until output is available.

For bends that are not at a 90-degree angle, you can select the radio button for Non-90 Degrees, enter the angle (0-180 deg), and the standard loss factor will be calculated.

Special Loss Types

The Bend also offers specialized loss types: the [3-K \(Darby\) method](#) and [Equivalent Length](#).

Additional Options

Basis Area for Loss Model - By default, the loss will be referenced to the upstream pipe area. This can be changed using the Basis Area for Loss Model dropdown.

Note: In principle, the upstream and downstream pipe diameters should be the same. However, the Bend junction only uses the reference area specified under Basis Area for Loss Model, even if the physical flow is in the opposite direction. If you are working with a user-specified loss, your geometry may have different diameter pipes connecting to the elbow. In this case, you would purposely define different upstream and downstream pipe areas. However, you would still want to be careful that the custom loss factor you assign is referenced to the correct area that is specified in Basis Area for Loss Model.

Restricted Flow Area - Optional input that is used only for checking if cavitation exists in the Bend junction.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Bend Loss Model.....	274

Bend Loss Model

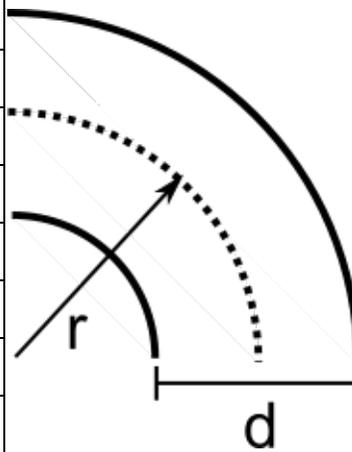
90 Degree Bends

The three [Bend](#) loss correlations are all for turbulent Reynolds numbers ([Crane 1988](#), A-29).

The K-factors for a **smooth flanged bend** with $r/D \geq 1$ are provided in Table 1.

Table 1: K values for 90 degree smooth flanged bends with $r/D > 1$

r/d	K
1	$20 f_T$
1.5	$14 f_T$
2	$12 f_T$
3	$12 f_T$
4	$14 f_T$
6	$17 f_T$
8	$24 f_T$
10	$30 f_T$
12	$34 f_T$
14	$38 f_T$
16	$42 f_T$
20	$50 f_T$



The **standard threaded elbow** is given by

$$K = 30 f_T$$

where f_T is the turbulent friction factor given in Table 2.

Table 2: Pipe friction factors used for Crane formulas

Nominal Size	Friction Factor f_T
1/2"	0.027
3/4"	0.025
1"	0.023
1 1/4"	0.022
1 1/2"	0.021
2"	0.019
2 1/2", 3"	0.018
4"	0.017
5"	0.016
6"	0.015
8"-10"	0.014
12"-16"	0.013
18"-24"	0.012

The **Mitre bend** is given by

$$K = 60 f_T$$

Non-90 Degree Bends

The two non-90 degree bend loss correlations are for turbulent Reynolds numbers ([Crane 1988](#), A-29).

A **smooth flanged bend** with $r/D \geq 1$ is given by

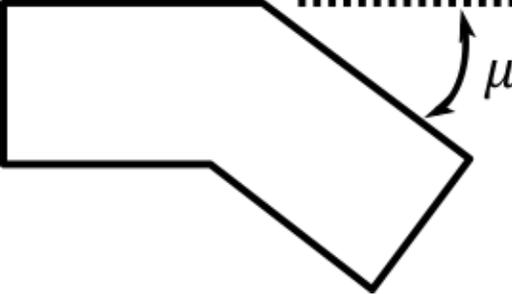
$$K_B = (n - 1) \left(0.25 \pi f_T \frac{r}{D} + 0.5K \right) + K$$

where n is the number of 90 degree bends, K is the loss factor for one 90 degree bend (Table 1), and f_T is given by Table 2.

The **Mitre bend** is given by Table 3.

Table 3: K values for Mitre bends

μ (deg.)	K
0	$2 f_T$
15	$4 f_T$
30	$8 f_T$
45	$15 f_T$
60	$25 f_T$
75	$40 f_T$
90	$60 f_T$



Smooth Flanged Bends, $r/D < 1$

The smooth flanged bend with $r/D < 1$ loss factor uses a correlation applicable for turbulent Reynolds numbers ([Miller 1990](#), 209)

The loss coefficient is calculated based on Miller's class 2 bend model, which is sufficient for general design purposes, and is given by

$$K_b = K_b^* C_{Re} C_0 C_f$$

C_0 and C_f are assumed to be 1 for class 2 bends. K_b^* is given by the nomograph on page 207 of Miller.

For $r/D > 0.7$ or $K_b^* < 0.4$, C_{Re} is given by Miller Figure 9.3. Otherwise, C_{Re} is given by the equation below.

$$C_{Re} = \frac{K_b^*}{K_b^* - 0.2 (C_{Re} \text{ for } r/D = 1) + 0.2}$$

where the C_{Re} for $r/D = 1$ is also found from Miller Figure 9.3.

Related Topics

K Factor Loss Model.....	261
Area Change Loss Model.....	269
Orifice Loss Model.....	288
Screen Loss Model.....	336
Tee/Wye Loss Model.....	342
Valve Modeling Closures with K Factors.....	352
Bend.....	272

Branch

The Branch junction allows up to twenty-five connecting pipes. The Branch junction is the most flexible junction type for building network models that have flow splits. Other significant features are the ability to impose a flow source or sink at the junction and the ability to specify separate loss factors for each connecting pipe. The Branch junction must have at least two connecting pipes.

The Branch Properties Window follows the second of the two basic [Properties Window formats](#).

A table on the Loss Coefficients folder tab displays the connecting pipe information. The pipe table grows in size to accommodate up to twenty-five connecting pipes. For each connecting pipe, the pipe table lists the reference flow direction and up to two loss factors. The first loss factor is for physical flow out of the branch and into the pipe. The second is for physical flow into the branch and out of the pipe. The loss factors can be specified independently or left as zero. The distance of each connecting pipe (if any difference exists) from the Branch elevation can be entered in the pipe table.

The Branch junction also offers you the option of specifying a flow source or sink at the junction in the Optional tab. According to [AFT Fathom's convention](#), a source of flow into the system is defined as positive, while a flow out of the system (a sink) is defined as negative. Along with the flow rate, you must specify the temperature when heat transfer is modeled.

Transient

([XTS Module](#) Only) On the Transient tab, a transient source or sink flowrate can be entered. For more information on transient data, including event transients, see [Junction Transient Data](#).

Related Topics

Junctions	248
General Junction Information	249
Junction Loss Models	261

Check Valve

The Check Valve junction type requires two connecting pipes. This junction type allows you to model valves that prevent reverse flow.

The Check Valve Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipes in a fixed format. The Check Valve junction does not have an explicit flow direction, but adopts a flow direction from the connecting pipes.

AFT Fathom assumes that the check valve is initially open. If the flow solution indicates that the specified Forward Velocity to Close is not met, AFT Fathom closes the valve and solves the flow distribution again. By default, the Check Valve will close at a velocity of 0. If a negative velocity value is specified, some amount of reverse flow will be allowed to occur before the valve closes. A positive velocity value means that the check valve will close while there is still forward flow through the valve.

Note: If the check valve closes during the simulation, it will not re-open. If there is an unsteady situation in which the check valve would continuously open and close, there may be unexpected results. It is good practice to close check valves with a Special Condition if it is known that they will be closed. This event can be seen in the Solution Progress window as "Re-running Solver because one or more special junction states changed..."

A check valve [Special Condition](#) can be either open or closed.

Specifying Losses

You specify the losses for the Check Valve on the Loss Model tab in the Properties window. When you select Resistance Curve, the Check Valve Properties window makes additional features available. Using these features, you can input Check Valve pressure drops that vary with flow. To enter these factors, you can specify polynomial constants, [fit a curve](#) to available data, or use interpolated x-y data. When a variable loss is specified, AFT Fathom dynamically modifies the loss factor in the Solver to agree with your input. You can choose any of the optional flow and pressure parameters provided; you can also specify the most convenient units.

An optional input in the Check Valve Properties Window is the Restricted Flow Area. This parameter describes the effective area restriction in the valve. This is done for the purpose of assessing cavitation. In most cases, information on the size of the restricted area must be obtained from the manufacturer or from test data.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Kv and Cv Loss Model.....	263
Check Valve closed at Junction X.....	785

Control Valve

Control Valve junctions are always internal to the system, with two connecting pipes. This junction type allows modeling of valves that offer special pressure or flow control characteristics at a location in the pipe system.

The Control Valve Properties Window follows the first of the [two basic Properties Window formats](#), displaying the connecting pipes in a fixed format. The Control Valve junction does not have an explicit flow direction, but adopts a flow direction from the connecting pipes.

Control Valve Types

You can model four types of control valves: Pressure Reducing Valves (PRV), Pressure Sustaining Valves (PSV), Flow Control Valves (FCV), and Pressure Drop Control Valves (PDCV). Loss information for a control valve is not required, because control valves are dynamic devices that change their

geometry in response to the pipe system behavior. The loss that results is that required to maintain the control parameter. You can, however, specify the full open loss. This is the loss that will occur should the valve fail to a full open state.

- A **PRV** is a device that controls the pressure in a pipe system. The PRV maintains a constant control pressure downstream of the junction as long as the upstream pressure exceeds the control pressure. If the upstream pressure is lower than the control pressure, the ability to control pressure is lost.
- A **PSV** is similar to a PRV in that it controls pressure in a pipe system. While the PRV maintains a constant downstream pressure, the PSV maintains a constant upstream pressure. If the downstream pressure rises higher than the control pressure, the ability to control pressure is lost.
- A **FCV** is a device that maintains a constant flow rate in a pipe system. By setting the junction to an FCV type and entering a flow rate, the junction will limit the flow through the connecting pipes to be equal to the control flow rate. The FCV can lose its ability to control flow when the pressure drop across it becomes zero or backward flow begins.
- A **PDCV** is a device that maintains a constant (stagnation) pressure drop. For this option, the valve is never allowed to not meet its setpoint. An indicator that an unrealistic pressure drop has been demanded is a failure to obtain a converged solution.

PRV/PSV Static vs. Stagnation Pressure

The control pressure for a PRV or PSV can be either [static or stagnation](#). The default selection is static, as this is the most frequent application in industry.

Action if Setpoint not Achievable

- **Always Control (Never Fail)** - It is possible that a control valve cannot achieve a given set point. By default this option is on - to change this, uncheck the box and click Set as Default. For additional information, see [Control Valve Setpoint Not Achievable](#).
- **Use Default Actions** - The behavior of the control valve when it cannot meet its setpoint can be modified by unchecking this option. The default behaviors represent the normal behaviors for the standard valve types.

Special Conditions

Control Valves have two [Special Conditions](#): to open fully and not control, or to close.

Open Percentage Table

The Optional tab allows data to be entered for special control valve characteristics. Specifically, the valve C_v and Flow Area can be specified vs. the Open Percentage of the valve. This data does not affect AFT Fathom's flow solution. However, engineers frequently desire to know the valve's open percentage during operation to ensure it meets design requirements. The Valve Summary (in the Output window)

always displays the valve C_v , and if data is specified for open percentage and flow area it will also display open percentage and flow area at the operating point. Tabulated data can be pasted or imported to the table by using the Edit Table menu to the right of the table. A graph of the open percentage, C_v , and flow area data can be displayed by selecting Show Graph to the right of the Open Percentage Data table.

A C_v vs Open Percentage table can be generated quickly for common C_v vs Open Percent profile types by choosing the Create C_v /Open %... button next to the table. This then opens a window in which a curve can be defined using one of the following options:

- **Transferred** - If a table has already been defined on the Optional tab of the Properties window, it can be copied over
- **User Defined** - Data can be entered directly into the table, or imported from existing data
- **Linear** - A linear curve is generated based on the user defined full open C_v and percent increment
- **Equal Percentage** - Generated as an [equal percentage](#) curve
- **Pre-Defined** - Pre-defined curves for several common valve C_v vs Open Percent profiles can be chosen and adjusted for a defined full open C_v value based on data from [Val-Matic 2009](#).

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Kv and Cv Loss Model.....	263
Control Valve Setpoint Not Achievable.....	280
FCV/PRV/PSV could not control pressure - Pressure was added to achieve control flow/pressure.....	773
FCV could not control flow at Junction X.....	777
PRV could not control pressure at Junction X.....	781
PSV could not control pressure at Junction X.....	781
Default control valve actions at Junction X used when setpoint not achievable because analysis is transient.....	785
Control valve open percentage outside valid range of 0-100% at Junction X.....	777

Related Examples

Control Valve.....	856
Pump Sizing and Selection with Flow Control Valves.....	882
Cooling System.....	895
Pump Sizing and Selection with Flow Control Valves - GSC.....	1001

Control Valve Setpoint Not Achievable

It is possible that the control valve cannot achieve the specified control setpoint. There are several actions the valve can take in such cases. By default, the Always Control (Never Fail) feature is selected. This option is physically unrealistic but useful for modeling purposes. The reason this option is physically

unrealistic is that the valve can add pressure to maintain control, thus acting like a pump. If this occurs, the user is clearly [warned in the output](#). The Always Control option is useful because it gives a clear indication to the user of how far away the valve is from being able to control. The pressure added to control can be seen in the Warnings section, or in the Valve Summary table on the Output window.

Flow Control Valves

For flow control valves, other options for valves which cannot achieve control are to have the valve Open Fully or Close. By default the valve fully opens. Once the control valve loses control, whether open or closed, AFT Fathom will solve the model again.

Pressure Control Valves

Pressure control valves are more complicated. These valves have two potential actions when control cannot be achieved. The first action is when the control cannot be achieved due to insufficient upstream pressure. The second is due to excessive downstream pressure. The default behavior for pressure reducing valves is to Open Fully due to insufficient upstream pressure, and close due to excessive downstream pressure. The defaults for pressure sustaining valves are the opposite. You can override the default behavior by clearing the Use Default Actions selection and specifying the desired actions.

AFT Fathom will determine if the valve can maintain control based on the Loss When Fully Open value. If this fully-open pressure drop is greater than the pressure drop which balances the system, the control valve can't achieve its setpoint. In other words, the control valve can't open wide enough (becoming less restrictive) to achieve control, and will revert to using the Loss When Fully Open specified by the user. If you enter data in the Open Percentage table on the Optional tab, you can select to have the data taken directly from the table for the 100% open case. You can also choose to not have any losses associated with the valve in the full-open state by selecting None.

Related Topics

Control Valve.....	278
FCV/PRV/PSV could not control pressure - Pressure was added to achieve control flow/pressure.....	773
FCV could not control flow at Junction X.....	777
PRV could not control pressure at Junction X.....	781
PSV could not control pressure at Junction X.....	781
Default control valve actions at Junction X used when setpoint not achievable because analysis is transient.....	785

Dead End

The Dead End junction must always have only one connecting pipe. This junction type is one way to define flow in a pipe as truly zero. Use of this junction allows AFT Fathom to skip the flow solution, since the flow is known to be zero.

The Dead End Properties window follows the first of the [two basic Properties window formats](#), displaying the connecting pipes in a fixed format. The only required information is the connecting pipe and the elevation.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261

General Component

The General Component junction type is internal to the piping system and requires two connecting pipes. This junction type allows you to model the irrecoverable loss that occurs through equipment that may not be explicitly represented on the [Toolbox](#) and to specify loss factors as a function of a flow parameter.

The General Component Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipes in a fixed format. The flow direction through the junction is determined by the defined directions of the connecting pipes.

When you select Resistance Curve, the General Component Properties Window makes additional features available. You can specify polynomial constants, [fit a curve](#) to available data, or use interpolated x-y data.

When a variable loss is specified, AFT Fathom modifies the loss factor in the Solver to agree with the solution. You can choose any of the optional flow and pressure parameters provided, and you can specify the most convenient units.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Resistance Curve Loss Model.....	264
Troubleshooting Resistance Curves.....	801
Heat Exchanger.....	282

Heat Exchanger

The Heat Exchanger junction type is internal to the piping system and requires two connecting pipes. This junction type allows you to model the irrecoverable loss that occurs through heat exchanger equipment. You also have the ability to specify loss factors as a function of a flow parameter.

The Heat Exchanger Properties window follows the first of the [two basic Properties Window formats](#), displaying the connecting pipes in a fixed format. The flow direction through the junction is determined by the defined directions of the connecting pipes.

When you select Resistance Curve, the Heat Exchanger Properties Window makes additional features available. Using these new features you can input heat exchanger pressure drops that vary with flow. To

enter these factors, you can specify polynomial constants, [fit a curve](#) to available data, or use interpolated x-y data.

When a variable loss is specified, AFT Fathom modifies the loss factor in the Solver to agree with the solution. You can choose any of the optional flow and pressure parameters provided, and you can specify the most convenient units.

Tube Configuration Model

The Heat Exchanger junction offers a special model for tube side pressure drop. The tube configuration model allows the user to specify tube properties and quantities, scaling and frictional characteristics, multiple passes, blocked tubes and K factors for turnarounds. The pressure drop is then calculated based on this data.

Heat Transfer

When [heat transfer](#) is modeled, the Thermal Data tab becomes active. See [Heat Exchanger Heat Transfer](#) to read about how this is modeled, and how to specify various configurations.

Special Conditions

The [Special Condition](#) for a Heat Exchanger junction functions differently than other junctions. Here you can turn off the heat transfer for the heat exchanger.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Resistance Curve Loss Model.....	264
Troubleshooting Resistance Curves.....	801
General Component.....	282
Heat Exchanger Heat Transfer.....	283

Related Examples

Freon Delivery System with Heat Transfer.....	888
Beginner - Cost Calculation.....	908

Heat Exchanger Heat Transfer

There are eleven different heat exchanger models available in AFT Fathom.

- NTU-Effectiveness Models
 - Counter-Flow
 - Cross Flow, Both Fluids Unmixed
 - Cross Flow, Secondary Fluid Mixed
 - Cross Flow, System Fluid Mixed
 - Parallel
 - Shell & Tube, 1 Shell Pass, Mult. of 2 Tube Passes
- Directly specified thermal property
 - Controlled Downstream Temperature
 - Enthalpy Stagnation Rise
 - Specified Heat Rate In Constant
 - Specified Heat Rate In vs Flow
 - Temperature Stagnation Rise

The image shows a software dialog box titled "Heat Exchanger Properties". It has several tabs: "Loss Model", "Thermal Data", "Optional", "Design Alerts", "Notes", and "Status". The "Thermal Data" tab is active. At the top, there are fields for "Number" (1), "Name" (Heat Exchanger), "Library Jct.", and "Copy Data From Jct...". To the right, there are fields for "Upstream Pipe" (None), "Downstream Pipe" (None), "Elevation", "Inlet" (feet), and "Outlet" (checked "Same as Inlet"). On the far right are buttons for "OK", "Cancel", "Jump...", and "Help". Below the tabs, the "Thermal Model" is set to "Counter-Flow". There is a checkbox "Link to Heat Exchanger:" which is unchecked. The "Thermal Data" section contains several input fields with units: "Heat Flow Into System" (Btu/s), "Controlled Outlet Temperature" (deg. F), "Heat Transfer Area" (feet2), "Overall Heat Transfer Coefficient" (Btu/hr-ft2-R), "Secondary Fluid Data" (Flow Rate: lbm/sec, Specific Heat: Btu/lbm-R, Inlet Temperature: deg. F).

Figure 1: Heat Exchanger Thermal Data

All of the NTU-Effectiveness models require the same 5 parameters as input (shown in Figure 1), but calculate the resulting heat transfer slightly differently. The direct specification models only require that the associated property be entered.

Thermal results are displayed along with pressure results in the Heat Exchanger Summary in the output, as well as information on secondary fluid conditions and log mean temperature difference.

Thermal Link

When modeling convective heat exchangers (such as parallel or counterflow) two heat exchanger junctions can be linked thermally. A thermal link is when two [heat exchanger junctions](#), which are hydraulically isolated, can exchange heat with each other. Typical use is when the two heat exchangers represent two sides of the same physical heat exchanger. When thermal linking specified in a heat exchanger properties window, the user must specify the heat transfer area and overall convection coefficient. AFT Fathom will force the two heat exchanger junctions to use the same value.

This capability can also be used to model heat transfer between loops with different fluids (see [Fluid Groups](#)).

NTU-Effectiveness Method

The NTU-Effectiveness method is used when the fluid inlet and outlet temperatures for the heat exchanger are not known.

To start, the NTU-Effectiveness method requires that the maximum possible heat transfer be found. Theoretically, the maximum heat transfer would occur in a counterflow heat exchanger of an infinite length. In this scenario, the maximum possible temperature difference is the difference between the inlet temperature of the hot fluid and the inlet temperature of the cold fluid:

$$\Delta T_{max} = T_{h1} - T_{c1}$$

The minimum heat capacity rate can then be found by multiplying the mass flow rate by the specific heat for the hot fluid and the cold fluid:

$$\begin{aligned} C_h &= (\dot{m}C_p)_h \\ C_c &= (\dot{m}C_p)_c \\ C_{min} &= \text{smaller of } C_h, C_c \end{aligned}$$

The fluid with the higher heat capacity rate changes temperature more quickly along the length of the heat exchanger, but the fluid with the lower heat capacity rate actually sees the maximum possible temperature change. Based on this, the maximum heat transfer between the two fluids in the theoretical heat exchanger is the minimum heat capacity rate multiplied by the maximum temperature difference:

$$q_{max} = C_{min} (\Delta T_{max})$$

The effectiveness of a heat exchanger is the ratio between the actual heat transfer rate and the maximum possible heat transfer rate:

$$\epsilon = \frac{q}{q_{max}} \quad \text{where} \quad q = C_h (T_{h1} - T_{h2}) = C_c (T_{c2} - T_{c1})$$

If the effectiveness and the inlet conditions are known, the amount of heat transferred between the two fluids can be calculated without knowing anything about either fluid's outlet temperature:

$$q = \epsilon C_{min} (T_{h1} - T_{c1})$$

The key to the NTU method becomes calculating the effectiveness, which is a function of the heat capacity ratio, C , and the number of transfer units, NTU:

$$C = \frac{C_{min}}{C_{max}}$$

$$NTU = N = \frac{UA}{C_{min}}$$

Each heat exchanger configuration has a unique relation used to calculate the effectiveness. These become the fundamental equations for the six thermal models that use the NTU method:

$$\textit{Parallel Flow} \quad \epsilon = \frac{1 - e^{-N(1+C)}}{1 + C}$$

$$\textit{Counterflow} \quad \epsilon = \frac{1 - e^{-N(1-C)}}{1 - Ce^{-N(1-C)}}$$

$$\textit{Crossflow, Both Fluids Unmixed} \quad \epsilon = 1 - e^{\left(\frac{N-0.22}{C}\right)\left(e^{-CN^{0.78}} - 1\right)}$$

$$\textit{Crossflow, Fluid with } C_{max} \textit{ mixed} \quad \epsilon = \frac{1}{C} \left(1 - e^{-C(1-e^{-N})}\right)$$

$$\textit{Crossflow, Fluid with } C_{min} \textit{ mixed} \quad \epsilon = 1 - e^{\frac{1}{C}(1-e^{-CN})}$$

$$\textit{Shell \& Tube} \quad \epsilon = 2 \left\{ 1 + C + \sqrt{1 + C^2} \left(\frac{1 + e^{-N\sqrt{1+C^2}}}{1 - e^{-N\sqrt{1+C^2}}} \right) \right\}^{-1}$$

Related Topics

Heat Exchanger.....282

Related Examples

Freon Delivery System with Heat Transfer	888
Beginner - Cost Calculation	908

Jet Pump

The Jet Pump junction is internal to the system and requires two connecting pipes, representing the suction and the discharge. The supply flow is specified explicitly and modeled internally in the junction type.

The Jet Pump Properties window follows the first of the [two basic Properties Window formats](#), displaying the connecting pipes in a fixed format. The flow direction through the junction is determined by the defined directions of the connecting pipes.

The Jet Pump junction is useful for modeling jet pumps, also known as eductors. Jet pumps are devices which use a high pressure injected fluid to entrain a lower pressure fluid and induce motion, thus pumping the lower pressure fluid. The high pressure fluid is referred to as the supply fluid, the low pressure fluid is the suction fluid, and the combined mixture is the discharge fluid. If the supply and suction fluid are different, the variable physical properties model should be used (selected in [Analysis Setup](#) on the [Heat Transfer/Variable Fluids panel](#)). In such a case, the data for the jet pump will be best entered on a mass flow rate basis, which includes the effect due to density.

To describe the performance characteristics, data must be entered for the pressure ratios at maximum and minimum flow, as well as the flow ratio at maximum flow. Additionally, the supply fluid pressure and flow rates are required.

If the pipe flow model includes [heat transfer](#), a supply temperature is required.

More information on Jet Pump calculations can be found in [Karassik, et. al, 1986](#) on pages 4.2-4.9.

Related Topics

Junctions	248
General Junction Information	249
Junction Loss Models	261
Flow rate at drop off point at Junction X	778

Orifice

The Orifice junction type requires two connecting pipes, unless you specify it as an exit orifice, in which case only one connecting pipe is allowed. This junction type allows you to model the irrecoverable loss that occurs through a plate-type flow restriction. It also allows you to specify loss factors as a function of a flow parameter and an orifice loss for pipes with an area change.

The Orifice Properties window follows the first of the [two basic Properties Window formats](#), displaying the connecting pipes in a fixed format. The Orifice junction does not have an explicit flow direction, but adopts a flow direction from the connecting pipes.

Orifices that are located at an exit are specified as exit orifices. These orifices require an associated back pressure definition. Exit orifices can only be connected to a single upstream pipe. To specify an exit ori-

fice, select the Exit Orifice checkbox, then specify the exit pressure and, if a [heat transfer](#) fluid model is used, the exit temperature.

Orifice Types

AFT Fathom provides two standard orifice types: sharp-edged and rounded. These loss factors are calculated based on [correlations](#). In addition to the standard orifices that AFT Fathom provides, you may enter your own custom orifice loss factor that is consistent with the application.

For convenience, you can specify constant loss characteristics of an orifice as a discharge coefficient (C_d) or as a loss factor (K). The standard orifice geometries include a functional dependence on the upstream and downstream pipe areas, which may be different.

When you select Resistance Curve, the Orifice Properties Window makes additional features available. Using these new features, you can input orifice loss factors or pressure drops that vary with flow. To enter these factors, you can specify polynomial constants, [fit a curve](#) to available data, or use interpolated x-y data. When a variable loss is specified, AFT Fathom modifies the loss factor in the Solver (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system) to agree with the solution. Here you can specify any of the available optional flow and pressure parameters using your preferred units.

An optional input in the Orifice Properties window is the Restricted Flow Area. This parameter describes the effective area restriction in the orifice. This is done for the purpose of assessing cavitation. In most cases, information on the size of the restricted area must be obtained from handbook or test data.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Orifice Loss Model.....	288

Orifice Loss Model

Sharp-Edged Orifice with Identical Upstream and Downstream Diameters

The orifice loss factors are all for turbulent Reynolds numbers. The sharp-edged orifice shown in Figure 1, is given by the following ([Idelchik](#) 2007, 259):

$$K_{up} = \left(\frac{A_{up}}{A_{orifice}} \right)^2 \left[0.707 \left(1 - \frac{A_{orifice}}{A_{up}} \right)^{0.375} + \left(1 - \frac{A_{orifice}}{A_{down}} \right) \right]^2$$

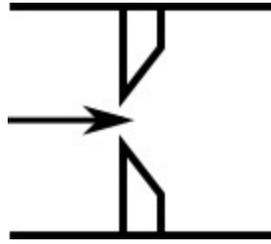


Figure 1: Sharp-edged orifice (in this case $A_{up} = A_{down}$)

Round-Edged Orifice with Different Upstream and Downstream Diameters

When the upstream and downstream pipe diameters are different for a round-edged orifice as shown in Figure 2, the round-edged orifice equation for different pipe areas is used and is given by the following (Idelchik 2007, 258)

$$K_{up} = \left(\frac{A_{up}}{A_{orifice}} \right)^2 \left[K' \left(1 - \frac{A_{orifice}}{A_{up}} \right)^{0.75} + \left(1 - \frac{A_{orifice}}{A_{down}} \right)^2 + 2 \left(1 - \frac{A_{orifice}}{A_{down}} \right) \sqrt{K' \left(1 - \frac{A_{orifice}}{A_{up}} \right)^{0.375}} \right]$$

where

$$K' = 0.03 + 0.47 * 10^{-7.7 r_{edge}/D_{orifice}}$$

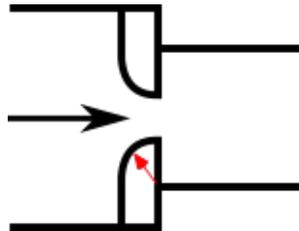


Figure 2: Round-edged orifice with different upstream and downstream pipe areas

Round-Edged Orifice with Identical Upstream and Downstream Diameters

When the upstream and downstream pipe diameters are the same for a round-edged orifice as shown in Figure 3, the **round-edged orifice equation for the same pipe areas** is used and is given by the following (Idelchik 2007, 262)

$$K_{up} = \left(\frac{A_{up}}{A_{orifice}} \right)^2 \left[1 - \frac{A_{orifice}}{A_{up}} + \sqrt{K'} \left(1 - \frac{A_{orifice}}{A_{up}} \right)^{0.75} \right]^2$$

where

$$K' = 0.03 + 0.47 * 10^{-7.7 r_{edge}/D_{orifice}}$$

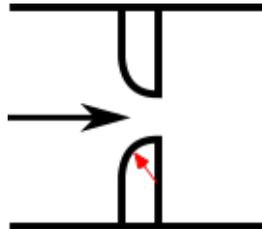


Figure 3: Round-edged orifice with the same upstream and downstream pipe areas

For other orifice configurations, see Chapter 4 of Idelchik's [Handbook of Hydraulic Resistance \(2007\)](#).

Orifice Discharge Coefficient

The relationship between orifice C_D and K is given by [Crane A-20](#):

$$K = \frac{(1 - \beta^2)(1 - \beta^4)}{C_D^2 \beta^4}$$

where β is the diameter ratio of the orifice diameter to the upstream pipe diameter, or

$$\beta = \frac{d_{orifice}}{d_{up}}$$

Related Topics

K Factor Loss Model.....	261
Bend Loss Model.....	274
Area Change Loss Model.....	269
Screen Loss Model.....	336
Tee/Wye Loss Model.....	342
Valve Modeling Closures with K Factors.....	352
Orifice.....	287

Pump

The Pump junction allows you to model the pressure added to a system by a pump, fan, or blowers (ignoring compression effects). Because all pumps are different, AFT Fathom does not provide any standard pump types. Pressure and flow data that describes the pump must be obtained from the manufacturer or from test data. To reduce repeated data entry, frequently-used pumps can be added to the [Component library](#).

Common Input Information

Like all junctions, pump junctions require a junction number, junction name, and inlet and outlet elevations. Similar to other junctions, pumps can have [Design Alerts](#) and Notes applied to them. They require two connecting pipes, unless the [Submerged Pump](#) option is enabled. Note that the direction of the connected pipes define what side of the pump is suction and which is discharge.

NPSH Reference Elevation

The NPSH Reference Elevation is the location where the pump manufacturer measured the NPSHR/A data, referenced to the same common elevation datum used by the rest of the model elevations. For most pumps this represents the centerline of the inlet suction flange. This input is required if the junction has different inlet and outlet elevations.

Pump Model

This tab requires the selection and definition of what type of pump the junction represents. There are three models available:

- [Centrifugal \(Rotodynamic\)](#) - Represents a typical centrifugal pump where the head added to the fluid is related to the flow through the pump via a [pump curve](#).
 - [Interstage Bleed/Takeoff Flow](#) - Represents a multiple stage pump that has some amount of flow removed between the upper and lower stages. Centrifugal (Rotodynamic) must be selected to enable this model.
- [Positive Displacement](#) - Represents a steady state approximation of a positive displacement pump. The flow is fixed to a constant value, and whatever head is necessary to obtain that flow will be supplied.
- [Pump As Turbine](#) - Represents a pump as a turbine which allows the pump to produce electrical power when the pump sees reverse flow.

Variable Speed

This tab allows the speed of the pump to vary either by a fixed amount, or dynamically. These options are only available for a centrifugal pump. A minimum speed can be entered to limit how much the speed is varied.

- **Fixed Speed (%)** - Directly modify the pump curve according to the [Affinity Laws](#). The Pump Curve entered in [Pump Configuration](#) is always assumed to be the 100% speed curve.
- **Controlled Pump (Variable Speed)** - Dynamically change the speed of the pump to meet a specified control setpoint. Instead of allowing the flow or head to vary with a fixed speed, the speed of the pump (and thus Pump Curve, according to the Affinity Laws) varies to meet a fixed flow or head. If modeling a suction or discharge pressure, control can optionally be enforced only if the value is above/below the setpoint.

Thermal

If heat transfer modeling is enabled, the Thermal tab will appear and heat added to the fluid by the pump due to inefficiency can be modeled. Three options are available:

- **Percent of Ideal Efficiency** - Enter the percentage of ideal energy required by the pump that goes into heat addition. Typically, this would equal the actual efficiency subtracted from 100%. For example, if the pump was 70% efficient, the entry here would be 30%.
- **Percent of Inefficiency** - Enter the percentage of the lost energy (i.e., the inefficiency) that goes into fluid heating. Typically, this would be 100%, which says that all inefficiency goes into heat.
- **User Specified Heat Rate** - Directly enter a heat load which goes into the fluid.

Slurry De-Rating

([SSL Module](#) Only)

Pump head degradation occurs when pumping solids. This is known as de-rating. See the [Slurry Pump De-rating](#) topic for more information.

Transient

([XTS Module](#) Only)

The pump junction allows modeling of transients to occur in the following pump parameters:

- Speed (if using a pump curve) for both Pumps and [Pump as Turbine](#)
- Flow rate (if modeling a fixed flow rate)
- Pressure/head (if modeling a fixed pressure or head rise)
- Control setpoint
- [Load Rejection](#) for Pump as Turbine

For more information on transient data, including event transients, see [Junction Transient Data](#).

Optional

There are several optional parameters that can be defined for a pump. Some of these are available to most junctions such as Initial Guesses, Display on Workspace, Design Factor, and Workspace Icon.

Also present are several pump junction specific options:

- **Special Condition**
 - **None** - The pump operates normally.
 - **Pump Off No Flow** - The pump is shut off and blocked. This is identical to a closed valve in Fathom.
 - **Pump Off With Flow Through** - The pump is shut off but allows flow to continue through. This is identical to a lossless connection in Fathom.
- **Number of Pumps at This Location** - Several pumps can be represented by one pump junction. This simplifies the model by reducing the number of pipes and junctions present, as well as automatically calculating the resulting composite pump curve based on the user-defined curve. When this option is selected the pump summary will show the total flow/head from the combined pumps.
- **Use [Viscosity Correction](#)** - Pump performance changes if the system fluid viscosity is different than the pump test fluid viscosity. Viscosity Corrections allow the user to account for these changes.
- **Additional [Efficiency Data](#)** - Power and efficiency information for each stage in a VFD-Motor-Pump assembly.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Centrifugal (Rotodynamic) Pump.....	294
Pump Configuration.....	296
Multiple Configurations.....	301
Pump Sizing.....	308
Positive Displacement Pump.....	314
Interstage Bleed/Takeoff Flow.....	309
Pump Efficiency Data.....	328
Viscosity Correction.....	313
Submerged Pump.....	326
Affinity Laws.....	324
Pump Reliability Factors.....	305
Junction X had reverse flow - Pump head could not be predicted.....	774
Junction X exceeded runout flow rate.....	779
Junction X %BEP (Y%) is above/below the POR range.....	779
Junction X flow (Y gal/min) is above/below the AOR range (Z gal/min).....	779
Junction X has insufficient pump NPSH/NPSP.....	779

Negative added pressure by pump at Junction X.....	779
Pump X has no efficiency curve data. Nominal efficiency was used in viscosity correction calculations.....	789
Pump X has no efficiency curve or nominal efficiency data.....	790
Pump X properties were outside of the valid range for the viscosity correction calculations. Results may be invalid.....	790
Poor Pump Curve Fits.....	800

Related Examples

Sizing a Pump.....	848
--------------------	-----

Centrifugal (Rotodynamic) Pump

Analysis Type

This topic pertains to the Pump Curve Analysis Option which represents the real behavior of a centrifugal pump. For Sizing Analyses, see [Pump Sizing](#).

Pump Curve

A Pump Curve is required to model a centrifugal pump. This is the typical Head or Pressure vs. Flow information that will be available from any pump manufacturer. Certain features also require the definition of an NPSHR curve, and Power or Efficiency curves.

To enter curves, click [Enter Curve Data](#). Once this data has been defined, previews of the applied curves can be seen here.

Trimmed Impeller

These values allow representation of different impeller trims without the need to input new curves.

- **Ratio as Percent** - This option assumes that the pump curve represents 100% trim. Entering different values adjusts the pump curve with the [Affinity Laws](#).
- **Actual Impeller** - Only available if [Multiple Configurations](#) are defined. In this case, a dimensional value for impeller size can be entered here.

Submerged Pump

Represents a pump with a fixed head or pressure at suction. See [Submerged Pump](#).

Check Valve at Discharge (No Backflow Allowed)

Model a [check valve](#) that can be considered part of the pump itself. If using the [XTS Module](#), a check valve junction will not close until the time step after the reverse flow occurs. However, when used as part of the pump junction it will never allow backflow at any time step.

- **Forward Velocity to Close** - The flow velocity, in the forward direction, that causes the check valve to close preventing any further flow. A negative value in this field indicates reverse flow through the pump.
- **Delta Head to Re-Open** - The differential head or pressure (upstream minus downstream) required to re-open the check valve.

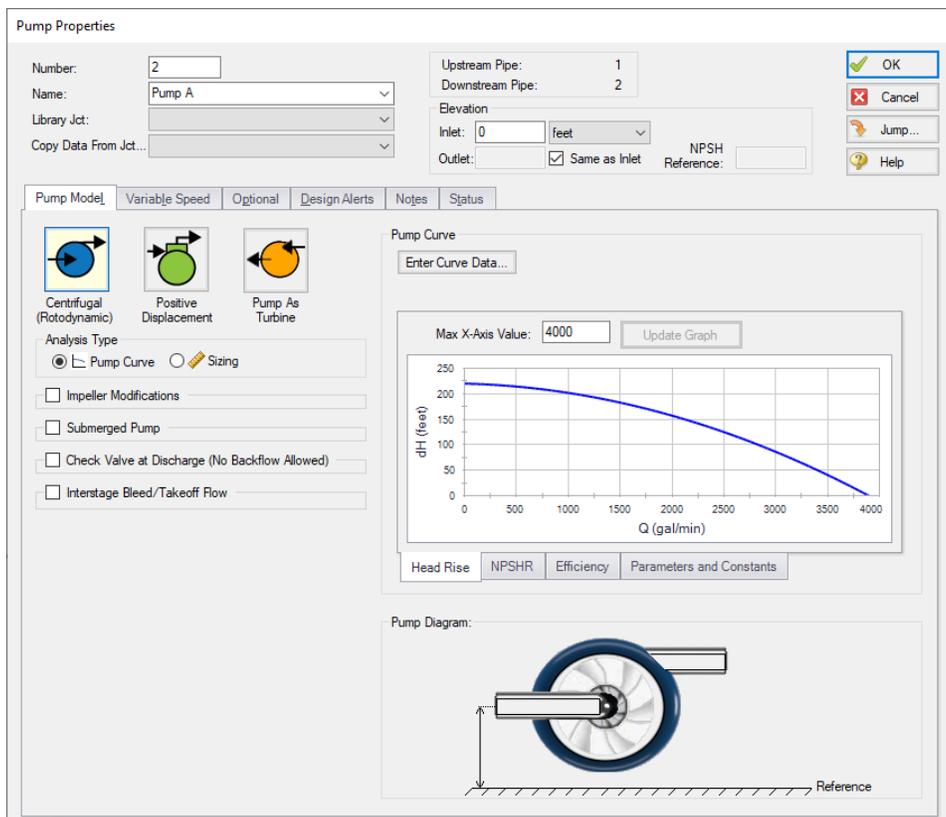


Figure 1: Centrifugal model with a pump curve defined

Interstage Bleed/Takeoff Flow

Represents a multiple stage pump that has some amount of flow removed between the upper and lower stages. See [Interstage Bleed/Takeoff Flow](#).

Related Topics

Pump.....	291
Pump Configuration.....	296
Multiple Configurations.....	301
Pump Sizing.....	308
Positive Displacement Pump.....	314
Interstage Bleed/Takeoff Flow.....	309
Pump Efficiency Data.....	328
Viscosity Correction.....	313
Submerged Pump.....	326
Affinity Laws.....	324
Pump Reliability Factors.....	305
Junction X had reverse flow - Pump head could not be predicted.....	774
Junction X exceeded runout flow rate.....	779
Pump X has no efficiency curve data. Nominal efficiency was used in viscosity correction calculations.....	789
Pump X has no efficiency curve or nominal efficiency data.....	790
Pump X properties were outside of the valid range for the viscosity correction calculations. Results may be invalid.....	790

Related Examples

Sizing a Pump.....	848
--------------------	-----

Pump Configuration

The pump configuration window is used to define the specific operation of a centrifugal pump. It is opened with the Enter Curve Data button from the [Pump Properties](#) Pump Model tab when Centrifugal is selected.



Figure 1: Enter Curve Data

Configuration Method

By default a pump is defined as a Simple configuration. This indicates that the pump junction can represent only one pump - defined by the curves in pump configuration.

If the Configuration Method is changed to Multiple, additional input becomes available on the right of the configuration window. This allows defining different operating speeds or impeller diameters for this pump, along with those configuration's specific curves. See a detailed description in [Multiple Configurations](#).

Pump Data

The first tab of the pump configuration controls the pump's most basic behavior - the pump curves.

Pump Configuration

Configuration Method
 Simple Multiple

Pump Data Configuration Data Composite Graph Performance Graph NPSH Graph Efficiency Graph

Raw Data: Optional Data

Parameter	Flow Q Volumetric Units	Head/Pressure dH feet	NPSHR feet	Efficiency/Power Efficiency Percent
1	0	45	8.4	
2	75			52
3	150	39.39	9.6	78
4	225			71
5	300	28.13	14.1	40
6				
7				
8				
9				
10				
11				
12				
13				
14				

Curve Fitting
 Curve Fit Type:
 Polynomial
 Interpolated X-Y Data
 Head Rise
 NPSHR
 Efficiency
 All None Invert
 Curve Fit Order
 2
 Generate Curve Fit Now

PUMP CURVE

NPSHR	
a	8.4
b	-0.002999992
c	7.333331E-05

EFFICIENCY	
a	-0.25
b	0.8926666
c	-0.002533333

$\Delta H = a + bQ + cQ^2 + dQ^3 + eQ^4$
 Reference Density ?
 No Correction Water @STP
 User Specified lbm/ft3

OK Cancel Help

Figure 2: Pump Data

A Pump can have three curves defined:

- **Flow vs. Head/Pressure** - Required. This is what is commonly referred to as the "pump curve" and defines how much head a pump can add to a fluid at any given flow rate.
- **Flow vs. NPSHR** - Optional. The NPSHR varies with flow and can be specified for a pump. Without specifying this curve, it is impossible to determine if cavitation may exist within the pump.

Note: NPSHR calculations require a reference elevation, which is defined in the Pump Properties window. For most pumps, the NPSH Reference elevation is the centerline of the suction of the pump. However, this should be checked with the pump manufacturer to determine where the NPSH curve is referenced.

- **Flow vs. Efficiency/Power** - Optional. The efficiency or power curve can be used to calculate the Best Efficiency Point (BEP), which is used for a variety of purposes. The definition of this curve also allows for the calculation of the actual power used or efficiency of the pump at its operating point. Without this curve, pump efficiency is assumed to be 100%.

After entering the raw values into the table, the data needs to be [curve fit](#).

If Mass Flow, Pressure, or Power are used in the curve definitions, [Reference Density](#) becomes an additional required input.

Configuration Data

This tab controls some additional behavior or input for a pump junction. Entering or changing values on this tab is optional.

The screenshot shows the 'Pump Configuration' dialog box with the 'Configuration Data' tab selected. The 'Configuration Method' is set to 'Simple'. The 'Best Efficiency Point (BEP)' section has 'Calculate from Efficiency/Power Data' selected. The 'End Of Curve Flow Rate (optional)' is set to an empty field with 'gal/min' units. The 'NPSHR Affinity Exponent At Operating Speed' is set to 'k = 2'. The 'Preferred Operating Region (POR)' section has 'None' selected, with a 'Specific Speed (US Customary)' dropdown set to '<=4500'. The 'Allowable Operating Region (AOR)' section has 'None' selected. The dialog includes 'OK', 'Cancel', and 'Help' buttons at the bottom.

Figure 3: Configuration Data

- **Best Efficiency Point (BEP)** - The Best Efficiency Point (BEP) by default is calculated from the entered efficiency or power curve. This is generally the preferred option as it best relates to the true meaning of BEP. A User Specified value can be entered if no efficiency or power curve is available, or if it is desired to force the BEP to a certain flow.
- **End Of Curve Flow Rate (optional)** - This value is used to create a [warning](#) in the output if it is exceeded. It does not affect the solution - the model will behave identically however this value is defined.
- **NPSHR Affinity Exponent At Operating Speed** - Pump curves - flow, head, and power - generally follow the [affinity laws](#). However, NPSHR does not always follow a standard relationship. The default is to assume the NPSHR varies with the square of the operating speed. If this is not the case, the value can be adjusted here.

$$NPSHR_{adjusted} = NPSHR_{original} * (Speed Ratio)^{Affinity Exponent}$$

$$Speed Ratio = \frac{Speed\ adjusted}{Speed\ original}$$

- **Preferred Operating Region (POR)** - The region where the pump operates most effectively. Percent of BEP values falling outside of the defined range will generate a warning in the model output. This range can be taken from [HI 9.6.3-2017](#) based on the specific speed of the pump, from [ANSI/API 610 \(2010\)](#), or can be defined by the user.
- **Allowable Operating Region (AOR)** - When an AOR is defined a warning will be created in the output when the user specified flow range is not adhered to. This is usually provided by the manufacturer.

Graphs

The following tabs display the curves that have been fit to the given data. If multiple curves are defined, the Composite Graph tab becomes available. The other graph tabs show the individual curves, along with points representing the raw data. It is a good idea to check that these graphs to confirm the curve fit is reasonable for any potential operating region.

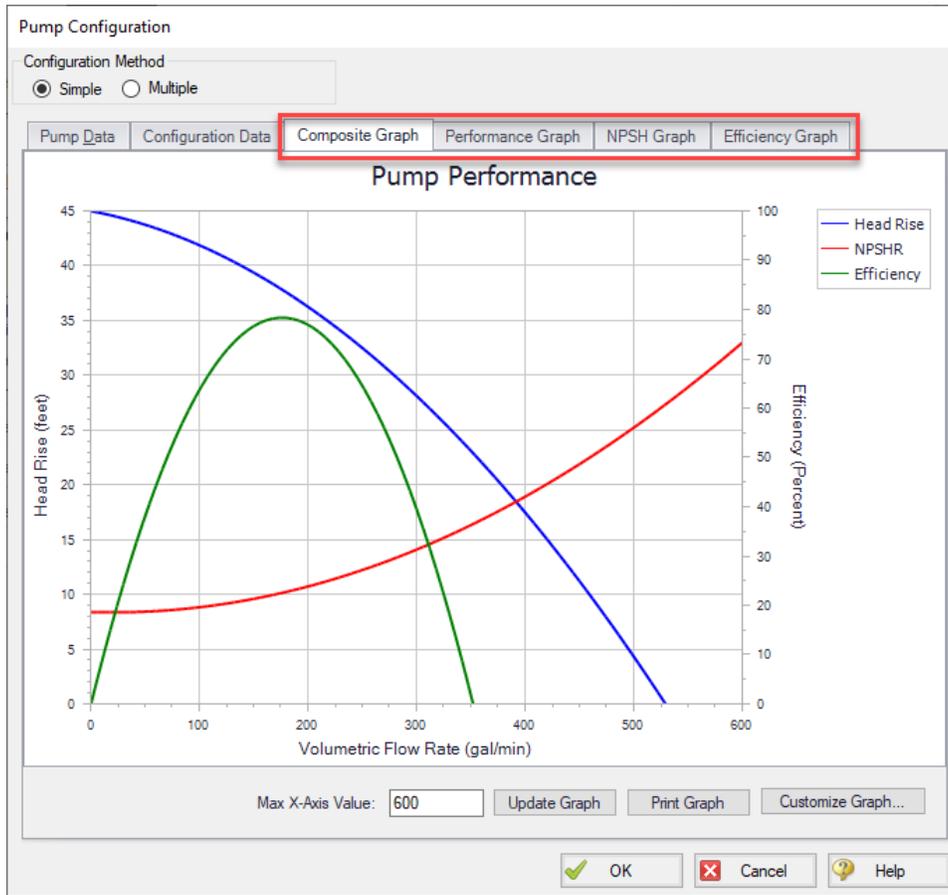


Figure 4: Composite Graph

Related Topics

Pump.....	291
Centrifugal (Rotodynamic) Pump.....	294
Multiple Configurations.....	301
Pump Sizing.....	308
Positive Displacement Pump.....	314
Interstage Bleed/Takeoff Flow.....	309
Pump Efficiency Data.....	328
Viscosity Correction.....	313
Submerged Pump.....	326
Affinity Laws.....	324
Pump Reliability Factors.....	305
Junction X %BEP (Y%) is above/below the POR range.....	779
Junction X flow (Y gal/min) is above/below the AOR range (Z gal/min).....	779
Junction X has insufficient pump NPSH/NPSP.....	779

Related Examples

Multiple Configurations

A single [pump junction](#) can contain information for multiple pump configurations - impeller trims and operating speeds. This is useful to compare different variations of a pump without needing to redefine a new pump junction for every case. The configuration of the pump can easily be changed without losing any information.

Multiple configurations also allow for an analysis of [pump reliability factors](#).

Defining Multiple Configurations

When opening the [pump configuration](#), change the Configuration Method from Simple to Multiple. When doing so, an additional user input panel appears on the right hand side of the window.

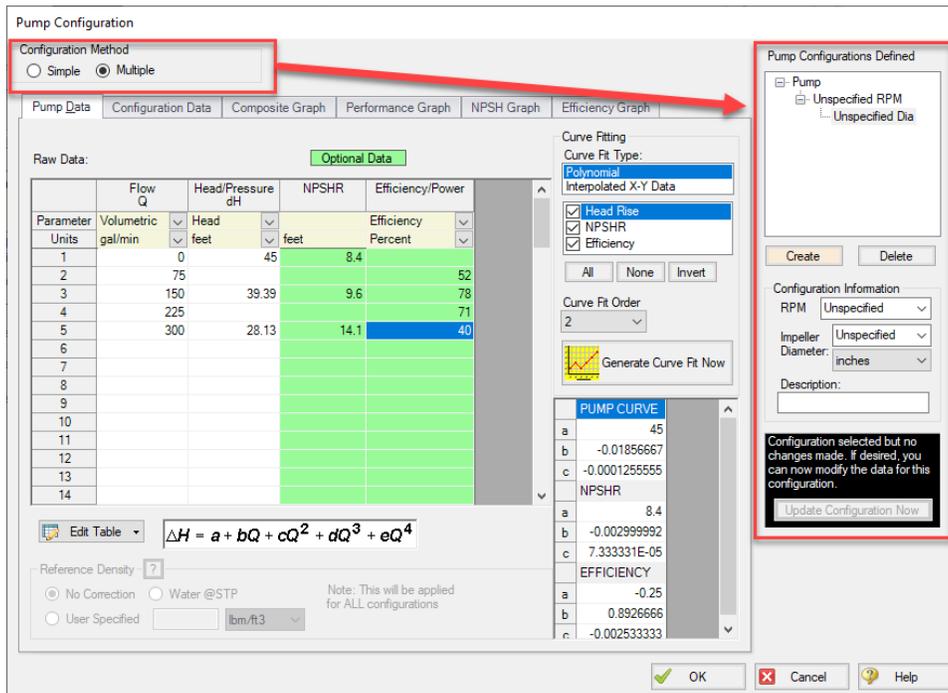


Figure 1: Multiple Configuration Method Selected

In order to make use of the multiple configuration feature, an RPM and Impeller Diameter under Configuration Information should be filled in. A Description can optionally be added. After modifying any information that affects the definition of this configuration the Update Configuration box turns yellow:

Junctions

Pump Configuration

Configuration Method
 Simple Multiple

Pump Data | Configuration Data | Composite Graph | Performance Graph | NPSH Graph | Efficiency Graph

Raw Data: Optional Data

Parameter	Flow Q Units	Head/Pressure dH feet	NPSHR feet	Efficiency/Power Percent
1	0	45	8.4	
2	75			52
3	150	39.39	9.6	78
4	225			71
5	300	28.13	14.1	40
6				
7				
8				
9				
10				
11				
12				
13				
14				

Curve Fitting
 Curve Fit Type:
 Polynomial
 Interpolated X-Y Data

Head Rise
 NPSHR
 Efficiency

All None Invert

Curve Fit Order
 2

Generate Curve Fit Now

PUMP CURVE

a	45
b	-0.01856667
c	-0.0001255555

NPSHR

a	8.4
b	-0.002999992
c	7.333331E-05

EFFICIENCY

a	-0.25
b	0.8926666
c	-0.002533333

$\Delta H = a + bQ + cQ^2 + dQ^3 + eQ^4$

Reference Density: No Correction Water @STP User Specified lbm/ft3

Note: This will be applied for ALL configurations

Pump Configurations Defined

- Pump
 - Unspecified RPM
 - Unspecified Dia

Create Delete

Configuration Information
 RPM: 3600
 Impeller Diameter: 12 inches
 Description:

Configuration completed but not yet updated. Click the 'Update Configuration Now' button to keep this data.

Update Configuration Now

OK Cancel Help

Figure 2: Update Configuration

Clicking Update Configuration Now adds the Configuration to the Pump Configurations listed in the top-right.

Junctions

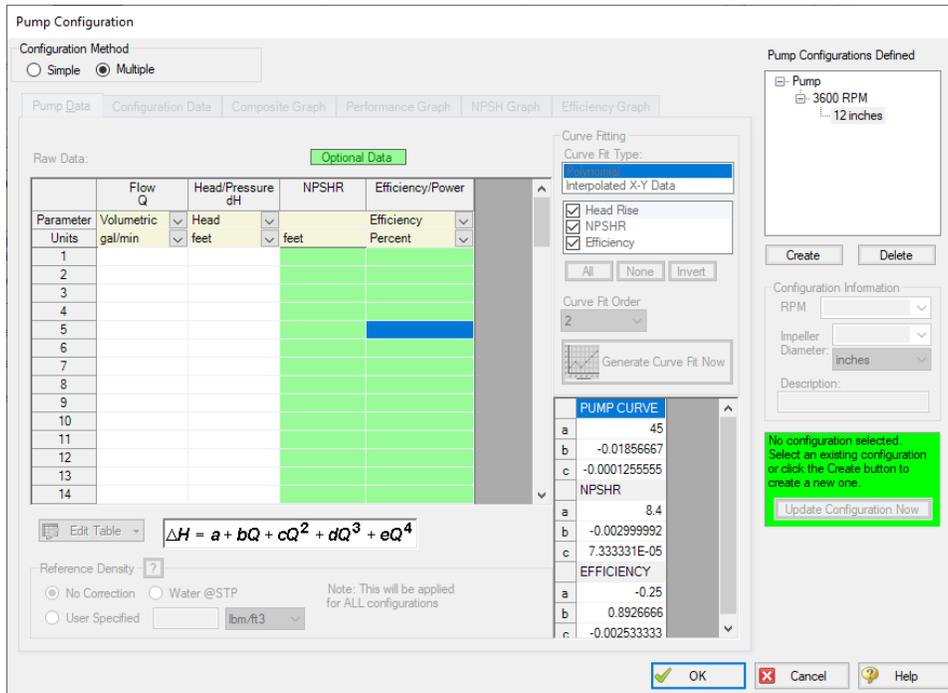


Figure 3: Current Pump Configurations

To create an additional configuration, click Create.

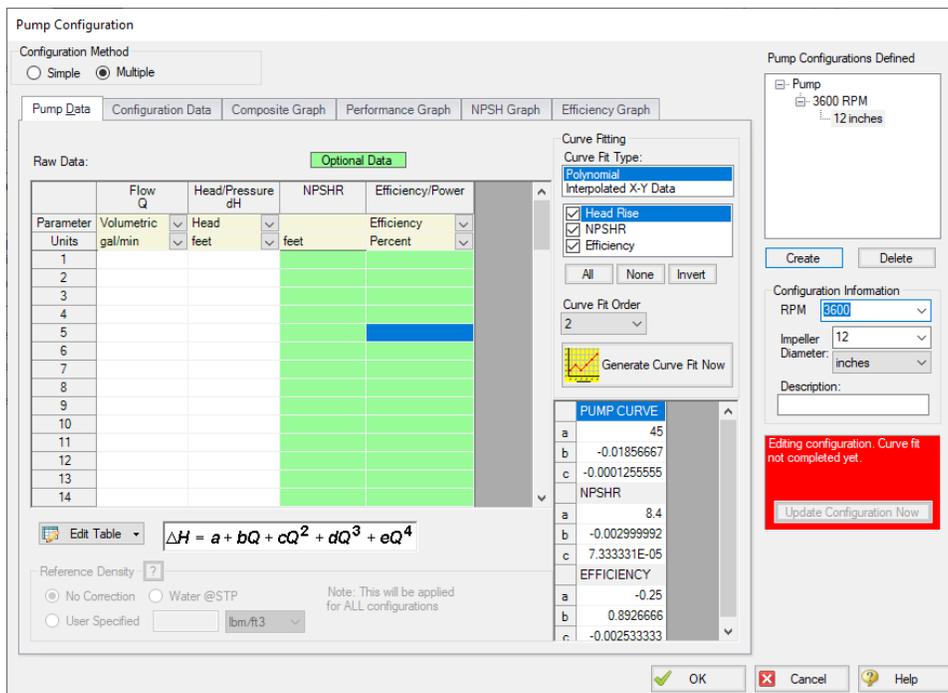


Figure 4: Creating a new Configuration

The pump curves must be created and curve fit, as with a single configuration pump. After completing this and clicking Update Configuration Now, the new configuration will be created and shown in the Pump Configurations area. Configurations can be edited by selecting them in this area.

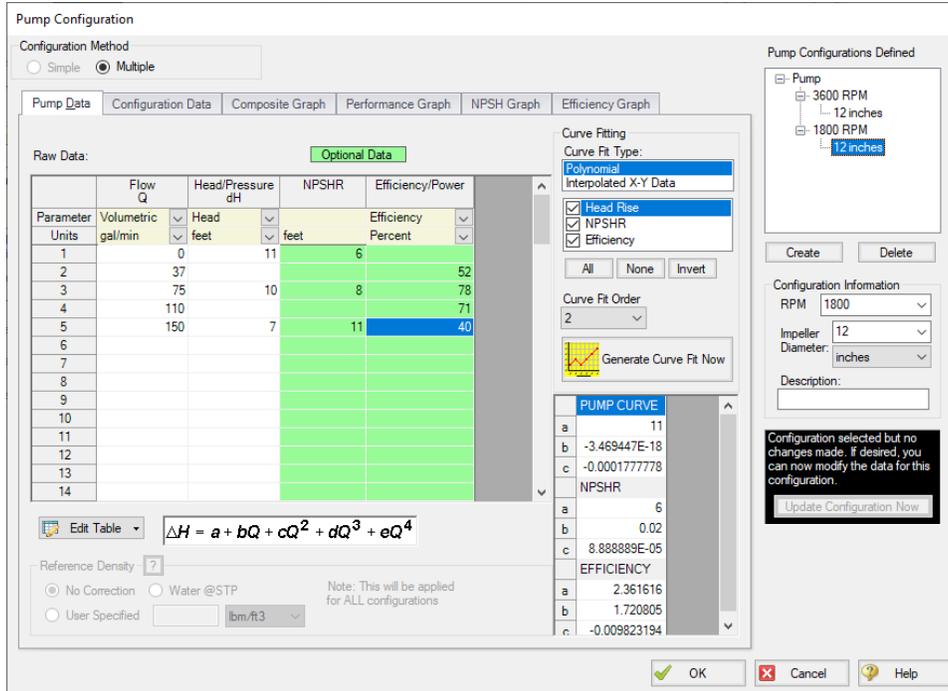


Figure 5: Editing a Configuration

Now that multiple configurations are available, they can be selected on the Pump Model tab:

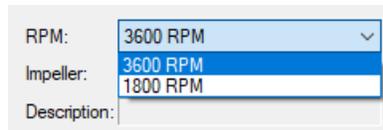


Figure 6: Selection of a Configuration

Related Topics

Pump.....	291
Centrifugal (Rotodynamic) Pump.....	294
Pump Configuration.....	296
Pump Sizing.....	308
Positive Displacement Pump.....	314
Interstage Bleed/Takeoff Flow.....	309
Pump Efficiency Data.....	328
Viscosity Correction.....	313

Submerged Pump.....	326
Affinity Laws.....	324
Pump Reliability Factors.....	305

Related Examples

Hot Water System.....	950
-----------------------	-----

Pump Reliability Factors

Reliability factors from [Bloch & Geitner \(1994\)](#) can be used to help assess the long-term reliability of pumps.

Setting up a Pump to Calculate Reliability

In order to calculate overall reliability, three pieces of pump data are needed:

- Efficiency vs. Flow
- Speed in RPM
- Actual, maximum, and minimum impeller diameters

Figure 1 shows the Pump Configuration window which is opened from the Pump Specifications window. In the table at the left of Figure 1, you can see where efficiency data is entered. The best efficiency point (BEP) is computed based on this data. If you would rather enter the BEP directly you can do so on the second tab Configuration Data. The BEP data is used to compute the reliability factor based on BEP. AFT Fathom calls it Reliability Factor BEP or **Reliability BEP**.

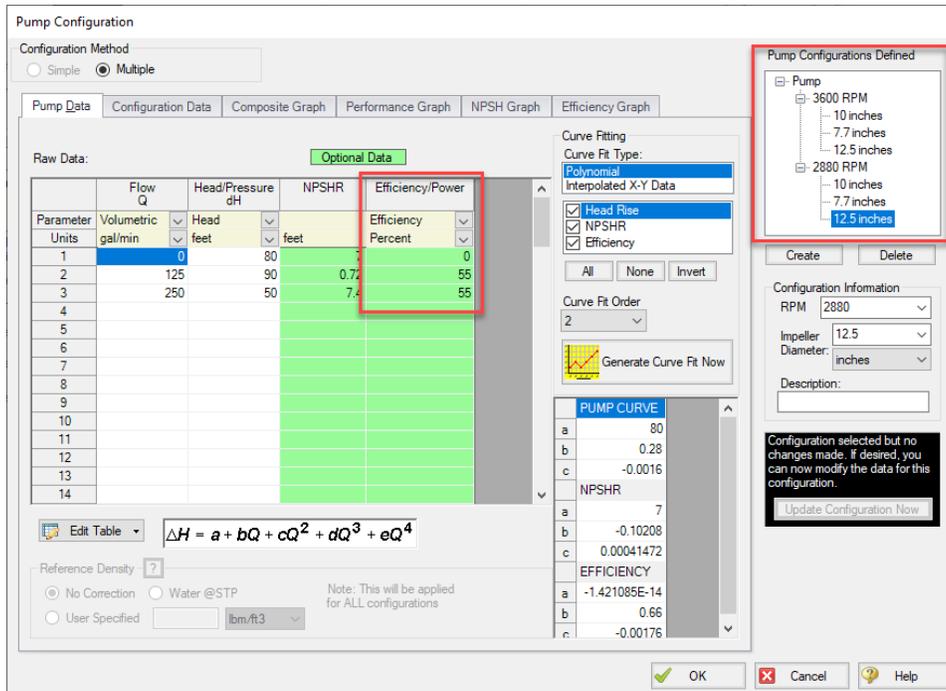


Figure 1: Pump Configuration window opened from Pump Specifications window

At the top left of Figure 1 you can see options for Configuration Method. To calculate the reliability factors for speed and impeller you need to use the [Multiple Configurations](#) option as shown in Figure 1. When you choose this, the window expands to the right and you can see the areas at the right of Figure 1 where RPM and impeller size are specified. Each RPM and impeller size combination is referred to as a pump configuration. To make the reliability factor calculations work correctly here is the data that must be entered for the pump:

- One configuration for the *maximum* impeller diameter must be entered
- One configuration for the *minimum* impeller diameter must be entered
- One configuration for the *maximum* speed in RPM must be entered

Using these three configurations, AFT Fathom searches all configurations for the pump and finds the maximum and minimum diameters to use for the impeller diameter based reliability factor which we call **Reliability Factor Diameter**. AFT Fathom also searches for the highest RPM value and assumes that as the design value. This value is then used for the **Reliability Factor Speed** calculation.

The **Reliability Factor Total** is simply the product of the BEP, Speed, and Diameter reliability factors.

Setting Pump Reliability in Output

The reliability factors can be selected for viewing using the Output Control window. Choose Output Control from the Tools menu or click the Output Control button on the toolbar.

Junctions

Choose the Pump button at the top of the Display Parameters tab, and from the list on the left select the reliability parameters you want to see in your output. Add them to the list on the right. When you run your model you will see the reliability factor output as in Figure 3.

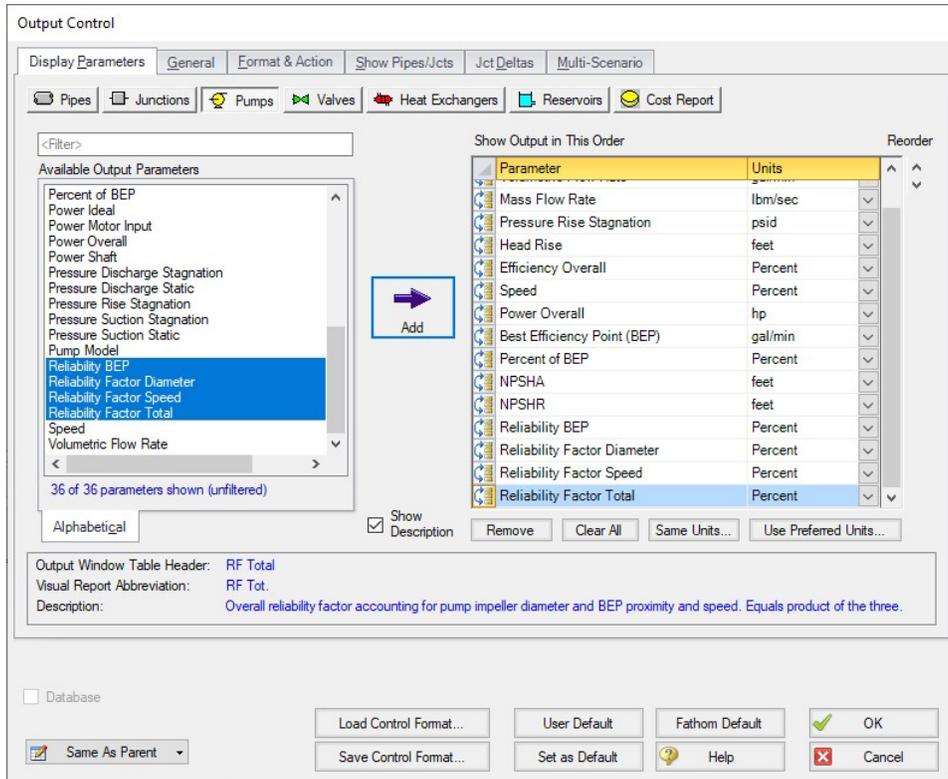


Figure 2: Output Control Window with Reliability Factors added

Jct	Results Diagram	Name	Vol. Flow (gal/min)	Mass Flow (lbm/sec)	dP (psid)	dH (feet)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (hp)	BEP (gal/min)	% of BEP (Percent)	NPSHA (feet)	NPSHR (feet)	RF BEP (Percent)	RF Diameter (Percent)	RF Speed (Percent)	RF Total (Percent)
2	Show	80% Impeller	113.2	15.73	38.07	87.93	58.16	100.0	4.323	150.0	75.49	33.17	N/A	96.14	85.59	20.00	16.46
5	Show	80% Impeller	-1,553.0	-215.71	27.71	64.00	N/A	100.0	N/A	120.0	N/A	45.13	N/A	0.00	89.66	46.40	0.00

Figure 3: Reliability Factors in the Pump Summary

Related Topics

Pump.....	291
Centrifugal (Rotodynamic) Pump.....	294
Pump Configuration.....	296
Multiple Configurations.....	301
Pump Sizing.....	308
Positive Displacement Pump.....	314
Interstage Bleed/Takeoff Flow.....	309

Pump Efficiency Data.....	328
Viscosity Correction.....	313
Submerged Pump.....	326
Affinity Laws.....	324

Pump Sizing

The [Centrifugal \(Rotodynamic\) Pump](#) Model offers the option of Sizing a pump. By entering a Pump Sizing Parameter, the corresponding operating conditions are calculated and displayed in the Pump Summary section of the [Output window](#) . By inputting a flow at a pump, determining the pump size requirements is straightforward. The [Walk Through Examples](#) have an example of using AFT Fathom to size a pump.

Pump Sizing with a Flow Control Valve

Using the sizing feature on a pump (fixing the flow rate) with a Flow Control Valve (which also fixes flow rate) downstream results in a Reference Pressure error - the system no longer has a unique solution. While the pump is being sized, the control valve must be temporarily defined as a Constant Pressure Drop Control Valve to avoid these errors. Read more about this in [Role of Pressure Junctions - Detailed Discussion](#).

Suggest Pump & Intelliquip™

Once you have sized a pump, you can use the Suggest Pump feature which is opened from the [Library menu](#) > Suggest Pump or the Output [Toolbar](#) to find an actual pump that meets the requirements. The Suggested Pumps window allows you to specify search criteria and to select the [libraries](#) you want to search. Search criteria includes pump head, NPSH, and BEP proximity.

If you or a colleague has previously saved various manufacturer pump models and performance curves to a [junction library](#) that you have connected in the [Library Manager](#), you can search this library to see if any saved pump curves meet the requirements. This is useful if an [Enterprise-Wide Network Library System](#) is set up for your company.

Click Search once the criteria and Source Library are defined (see below for instructions on using the Internet Library Source). Pumps that meet the criteria will be shown in the table below. The table can be sorted by proximity to operating head, BEP, or alphabetically. The Show Pump Curve button allows for quick viewing of the composite curve for a selected pump. The operating point is overlaid on the graph such that you can determine if the pump is a good fit. If the performance curve is determined to be appropriate for the model, the Apply Pump Data button can be used to open the Apply Pump Data window where one or more pump junctions can be updated to use the performance curve.

Intelliquip™ - In addition, the Internet Library Source allows you to pass operating point data to Internet hosted pump supplier websites which use Intelliquip™.

1. Select the Internet radio button under Source Library, and select Intelliquip from the dropdown menu.
2. Click Search to open the Intelliquip Search window.

3. Select Display Units and the Frequency of the electrical grid that the pump will operate on (60 Hz or 50 Hz).
4. Click Open Web Browser to be taken to the AFT Intelliquip landing page.
5. Select a pump supplier website.
6. Browse the catalog and adjust the search filters. Each supplier's website may operate differently and might require different data inputs to be adjusted.
7. Once a pump model is found to be appropriate, look for a download button and download the *.iqx file.
8. Return to the Intelliquip Search window in Fathom, click the radio button for Step 2, and click Open File(s).
9. Select the *.iqx file for the pump. The pump will now appear in the table.
10. Select the pump in the table, click Apply Pump Data, and update the desired junctions in the model. The Add to Library option allows you to save the pump curve to a connected library for future use.

For an example of pump sizing in Fathom, see the [Sizing a Pump](#) example.

Related Topics

Pump.....	291
Centrifugal (Rotodynamic) Pump.....	294
Pump Configuration.....	296
Multiple Configurations.....	301
Positive Displacement Pump.....	314
Interstage Bleed/Takeoff Flow.....	309
Pump Efficiency Data.....	328
Viscosity Correction.....	313
Submerged Pump.....	326
Affinity Laws.....	324
Pump Reliability Factors.....	305

Related Examples

Sizing a Pump.....	848
Heat Exchanger System.....	866
Pump Sizing and Selection with Flow Control Valves - GSC.....	1001
Cooling System - ANS.....	1176

Interstage Bleed/Takeoff Flow

Some multiple stage pumps can have flow removed at some intermediate point. For such pumps, the manufacturer provides two pump curves:

- **Overall Pump Curve** - The total dynamic head from the suction of the first stage to the discharge of the last stage *when the takeoff flow is zero*.
- **Takeoff Pump Curve** - The total dynamic head of the stages *before* the takeoff point.

Note: The **Overall Pump Curve** is an *effective* curve that describes the operation of all pump stages together. This curve cannot be directly used to determine the overall head when the takeoff flow is anything other than zero.

Defining the Model

First, place two pump junctions onto the Workspace, representing the upstream and downstream stages of the multistage pump.

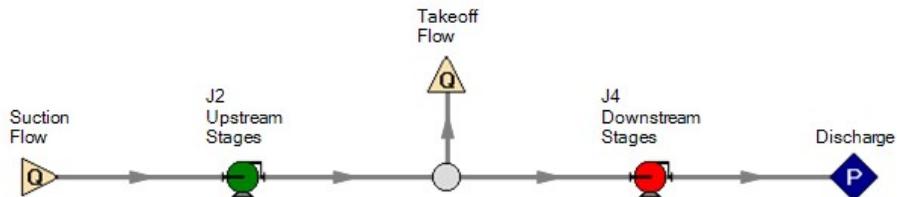


Figure 1: Example Workspace layout - color for emphasis only

Both pumps need to be defined as "Interstage Bleed/Takeoff Flow" as shown below. The "Upstream Stages" pump should have the Takeoff Curve entered. The "Downstream Stages" pump should have the Overall Pump Curve entered.

Keep in mind that the pipes connecting the two pumps **do not exist in reality**. For this reason, it is reasonable to make these pipes have as little impact as possible. This can be done by making them short and/or frictionless. Otherwise, there will be some nominal frictional head loss in the section where effectively none would exist in the real system.

Additionally, the two pumps need to be "connected" by setting the junction of the corresponding pump junction in this area.

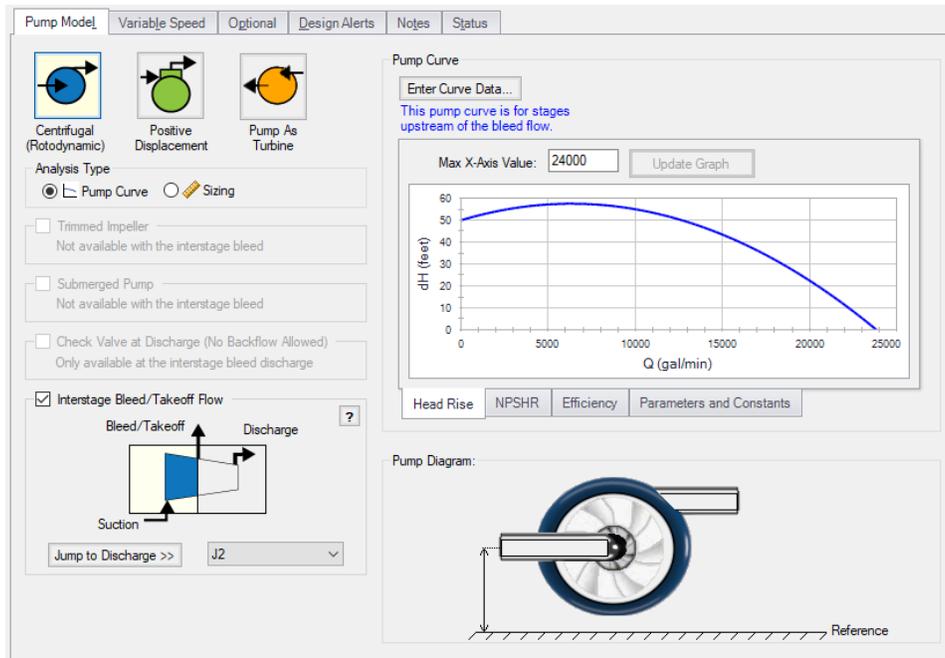


Figure 2: Interstage Configuration

Methodology

As can be seen in Figure 3, the Suction Flow is always equal to the sum of the Takeoff and Discharge Flows. Also important to note is that the Suction Flow is always equal to the discharge flow of the upstream stages - that is, adjusting the takeoff flow does not directly affect the behavior of the upstream stages.

From this, we can determine the head that is added by the upstream stages by examining the Takeoff Pump Curve at the Suction Flow. This is shown in Figure 4 as Upstream Stages Head.

Next, we need to determine the head contributed by the Downstream Stages. However, we do not know the pump curve for these stages! By subtracting the Takeoff Pump Curve from the Overall Pump Curve, we can determine the Downstream Stages Pump Curve. Knowing the Discharge Flow (Suction Flow - Takeoff Flow), we can now look up the Downstream Stages Head, as shown in Figure 4.

Adding these two values brings us to the Overall Effective head across the entire pump at the given Discharge Flow. This value will be smaller than the Overall Pump Curve value due to the head lost to the Takeoff Flow.

Similarly, adding the overall power calculated at each of the pump junctions will give the ideal Overall Effective power for the entire pump.

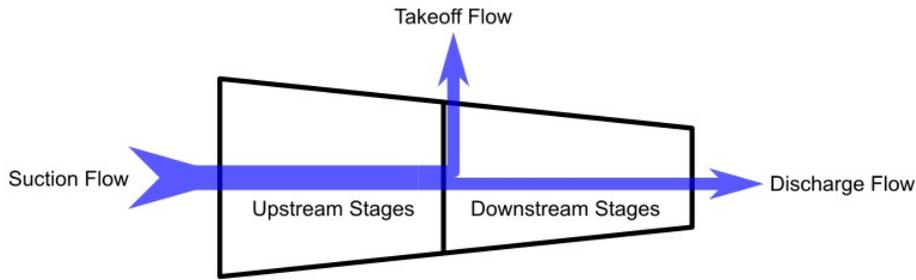


Figure 3: Interstage Diagram

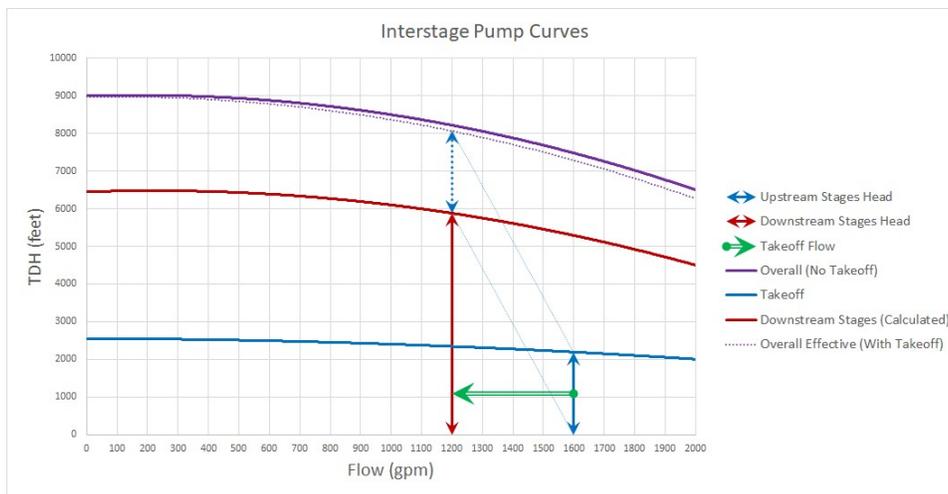


Figure 4: Interstage Curves

Related Topics

Pump.....	291
Centrifugal (Rotodynamic) Pump.....	294
Pump Configuration.....	296
Multiple Configurations.....	301
Pump Sizing.....	308
Positive Displacement Pump.....	314
Pump Efficiency Data.....	328
Viscosity Correction.....	313
Submerged Pump.....	326
Affinity Laws.....	324
Pump Reliability Factors.....	305

Viscosity Correction

Most pump curves are developed with pure water; however, pumping a fluid more viscous than water will result in a reduction of pump performance - even if the [fluid reference density](#) is the same as water. The Viscosity Correction feature employs the methods described in [ANSI/HI 9.6.7-2021 Rotodynamic Pumps - Guideline for Effects of Liquid Viscosity on Performance](#) to correct for losses in performance due to viscosity.

Viscosity Corrections should only be used if the pump curve entered into the Pump Properties window is a water curve, since this is the only situation covered by ANSI/HI 9.6.7-2021. Viscosity Corrections will effectively transform a water pump curve into a viscous pump curve. If the manufacturer provides a pump curve for your working fluid at your desired viscosity, then it should be entered directly and the Viscosity Correction feature should remain disabled, otherwise the effects would be double counted. If the manufacturer provides a pump curve based on your same working fluid, but at a different viscosity, then the Viscosity Correction feature should still remain disabled because unfortunately that case falls outside the scope of the ANSI/HI 9.6.7-2021 standard.

Correction factors referred to as C_H (for head rise), C_Q (for volumetric flow), and C_E (for efficiency) are defined and can be displayed in the Pump Summary section of the Output:

$$C_H = \frac{H_{viscous}}{H_{water}} \quad C_Q = \frac{Q_{viscous}}{Q_{water}} \quad C_E = \frac{\eta_{viscous}}{\eta_{water}}$$

To calculate the correction factors, the water BEP (head and flow at BEP) and rated pump speed are required.

- **Assume Pump is Operating at BEP** - This option is useful when the pump is in Sizing mode, and will assume that the head rise and flowrate calculated by the model are at BEP. This option is also useful when a water pump curve is entered but no efficiency data is present.
- **Use Water BEP**
 - **From Pump Configuration** - The water BEP flowrate can be determined from the pump curve data entered on the [Pump Configuration window](#). This option will be disabled if efficiency data is not present.
 - **User Specified** - Alternatively, the user can directly specify the water BEP flowrate.

Scope of Correction

It is important to keep in mind that, like all mathematical models, this standard is not universally applicable. Below are the primary requirements:

- Single or multistage centrifugal pump
- Essentially radial discharge with specific speed under 3000 (US units)
- Newtonian Fluid
- Kinematic viscosity between 1 and 4000 cSt

For detailed information regarding these limitations and the application of the standard, please consult ANSI/HI 9.6.7-2021.

Related Topics

Pump.....	291
Centrifugal (Rotodynamic) Pump.....	294
Pump Configuration.....	296
Multiple Configurations.....	301
Pump Sizing.....	308
Positive Displacement Pump.....	314
Interstage Bleed/Takeoff Flow.....	309
Pump Efficiency Data.....	328
Submerged Pump.....	326
Affinity Laws.....	324
Pump Reliability Factors.....	305
Pump X has no efficiency curve data. Nominal efficiency was used in viscosity correction calculations.....	789
Pump X has no efficiency curve or nominal efficiency data.....	790
Pump X properties were outside of the valid range for the viscosity correction calculations. Results may be invalid.....	790

Related Examples

Crude Oil Pipeline.....	943
-------------------------	-----

Positive Displacement Pump

Positive Displacement pumps can be used to provide a fixed flow for steady state analysis.

Parameter

- **Volumetric Flow Rate** - Fix the volumetric flow rate through the pump.
- **Mass Flow Rate** - Fix the mass flow rate through the pump.

Fixed Condition

- **Fixed Flow Rate** - The value and unit of the fixed flow amount.
- **Nominal Efficiency** - A fixed efficiency, useful to calculate [power requirements](#). This is an optional input.
- **Nominal NPSHR** - Net Positive Suction Head Required for the pump at the specified flow. This is an optional input.

Acceleration Head/Pressure Parameters

Acceleration head is the energy required to change the velocity of the fluid in the system from zero to some non-zero velocity. This requires additional energy beyond the amount required simply to overcome the frictional losses in a system. This is reflected when starting a centrifugal pump - the peak power requirement is higher than during steady operation. Similarly, positive displacement pumps are constantly accelerating and decelerating the fluid in the suction line. The value for acceleration head can be added to the Pump Summary via [Output Control](#), and is visible in the [Applied Standards](#) tab.

- **Type of Pump (C)** - The type of Positive Displacement Pump, for the purposes of calculated the acceleration head pump type coefficient, as described below. If no predefined types are applicable, a user specified value must be entered.
- **K** - The compressibility factor of the fluid, as described below.
- **Pump Speed** - The nominal RPM of the Positive Displacement Pump.

This suction line acceleration head can be calculated with the method defined in [ANSI/HI 6.1-6.5-2015](#). This method requires the user definition of pump speed, pump type coefficient, and a fluid compressibility factor. Suction line length and fluid velocity are also required and are calculated by AFT Fathom.

$$h_{acc} = \frac{lvnC}{Kg}$$

- **h_{acc}** - Acceleration head (ft)
- **l** - Length of suction line (ft)
- **v** - Fluid velocity in suction line (ft/s)
- **n** - Pump speed (rpm)
- **C** - Coefficient determined by type of reciprocal pump:
 - Simplex (Single-acting) = 0.628
 - Simplex (Double-acting) = 0.200
 - Duplex (Single-acting) = 0.200
 - Duplex (Double-acting) = 0.115
 - Triplex = 0.066
 - Quintuplex = 0.040
 - Septuplex = 0.028
 - Nonuplex = 0.022
- **K** - Factor representing relative compressibility of fluid. Some common compressibility factors are provided.
 - Hot Water = 1.4
 - Hot Oil = 2.5
- **g** - Gravitational constant (ft/sec²)

AFT Fathom calculates the total acceleration head by summing the calculated acceleration head for each individual pipe. The solver will attempt to automatically determine the pipes in the suction line by

tracing the flow upstream of the pump back to an [Assigned Pressure](#) or [Reservoir](#) junction. If at any point upstream of the pump there is more than one potential path (for example, two inflows merging together), this calculation will not proceed because the suction line is indeterminate. However, the flow can split into multiple outflows. As long as there is one continuous and unique inflow path to the pump, acceleration head can be calculated.

Note: Though the ANSI/HI 6.1-6.5-2015 standard states that the NPSHA should be corrected to account for acceleration head, Fathom will not adjust the NPSHA shown in the output. The user should subtract the acceleration head from the displayed NPSHA to comply with the standard.

Submerged Pump

Represents a pump with a fixed head or pressure at suction. See [Submerged Pump](#).

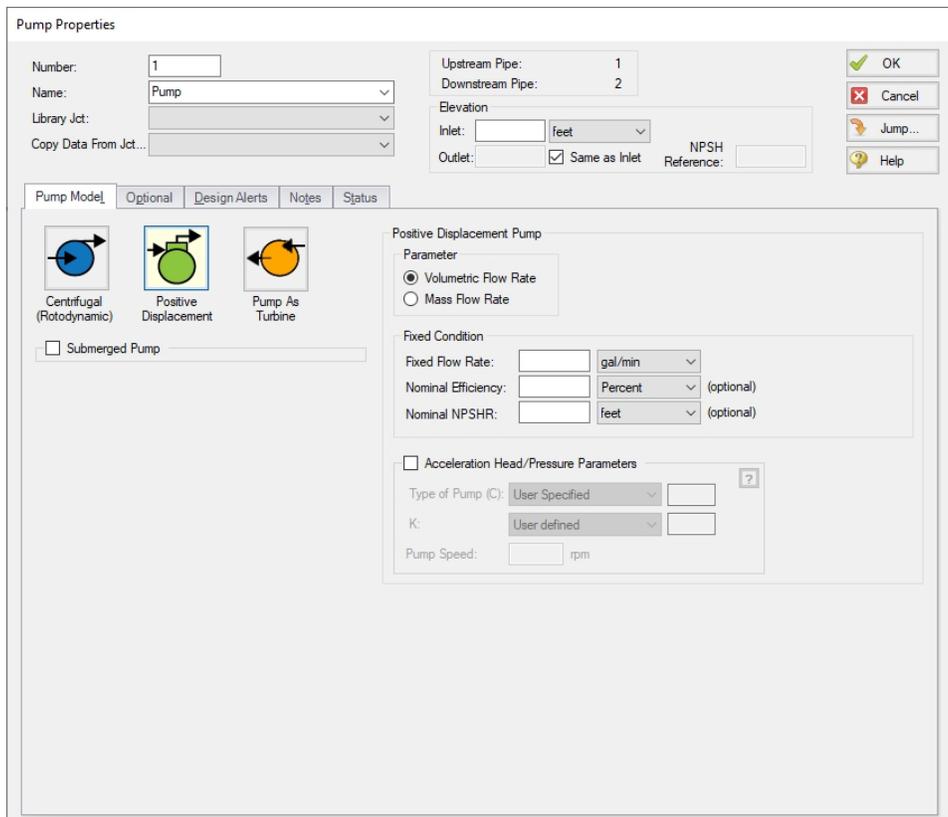


Figure 1: Positive Displacement Pump Model

Related Topics

Pump.....	291
Centrifugal (Rotodynamic) Pump.....	294

Pump Configuration	296
Multiple Configurations	301
Pump Sizing	308
Interstage Bleed/Takeoff Flow	309
Pump Efficiency Data	328
Viscosity Correction	313
Submerged Pump	326
Affinity Laws	324
Pump Reliability Factors	305

Pump as Turbine Overview

Running a pump as a turbine allows the pump to produce electrical power when the pump sees reverse flow. During reverse flow, the motor coupled to the Pump as Turbine (PAT) spins in reverse, causing the magnetic rotor to induce a current in the motor coils, turning the motor into a generator.

Note: The positive flow direction for the pipes connected to the PAT should be the direction in which flow will travel when the pump is operating as a turbine, rather than the direction in which flow will travel when the pump is operating as a pump. Thus, while the physical pump equipment will see reverse flow while operating as a turbine, the pump as a PAT will see positive flow.

Standard Turbine Curve

A Standard Turbine Curve is required to model a PAT and must include both head and power data. The head loss vs flow curve defines the head loss seen at the PAT while coupled to the motor and generating power. The power versus flow curve defines the power generated by the PAT while coupled to the motor and generating power.

To enter these curves, click Enter Curve Data. Previews of the applied curves can be viewed in the Standard Turbine Frame once the data has been defined.

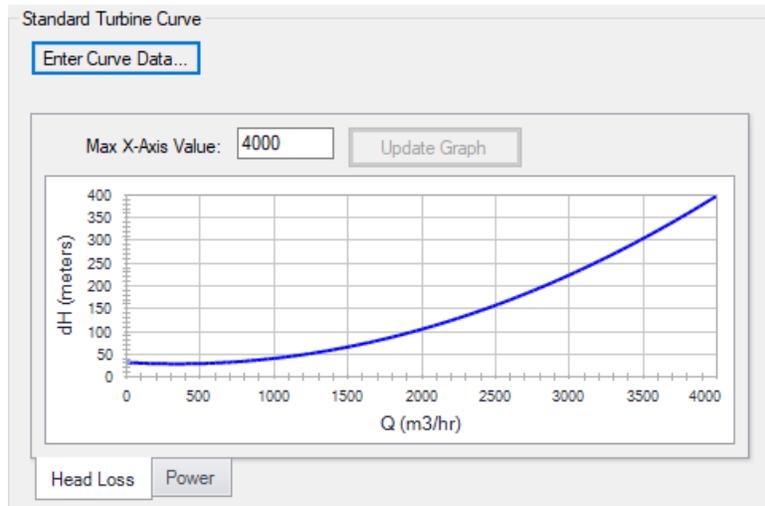


Figure 1: Standard turbine curve

Flow Directional Behavior

A visual representation of the modeled PAT is presented in the lower right of the properties window to provide immediate feedback on what the various options represent. In the PAT depicted in Figure 2, head loss and power information have been defined, and the PAT is in normal operation, with reverse flow seen at the pump.

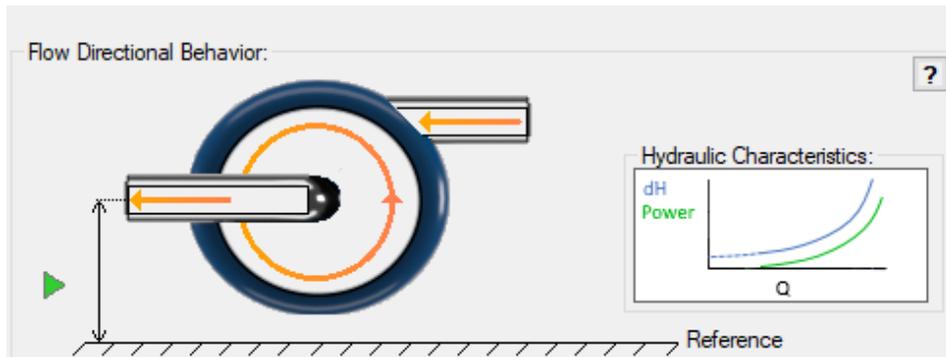


Figure 2: Flow Directional Behavior image for an example PAT

PAT Configuration

The Pump as Turbine Configuration window is used to define the specific operation of a Pump as Turbine. It is opened by clicking Enter Curve Data from the Pump Properties window on the Pump Model tab when Pump as Turbine is selected.

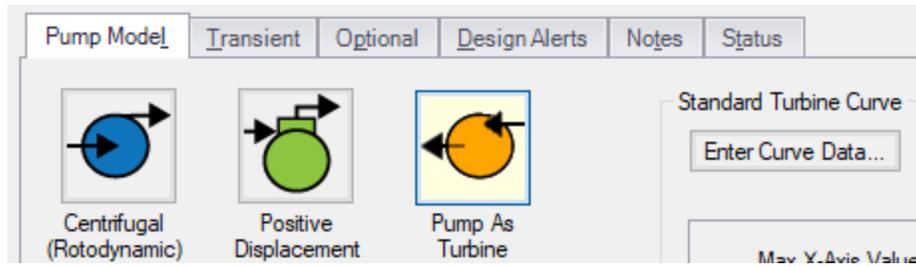


Figure 1: Enter Curve Data on the Pump Model tab

Turbine Data

The first tab of the PAT configuration defines the PAT's most fundamental behavior - the turbine curves.

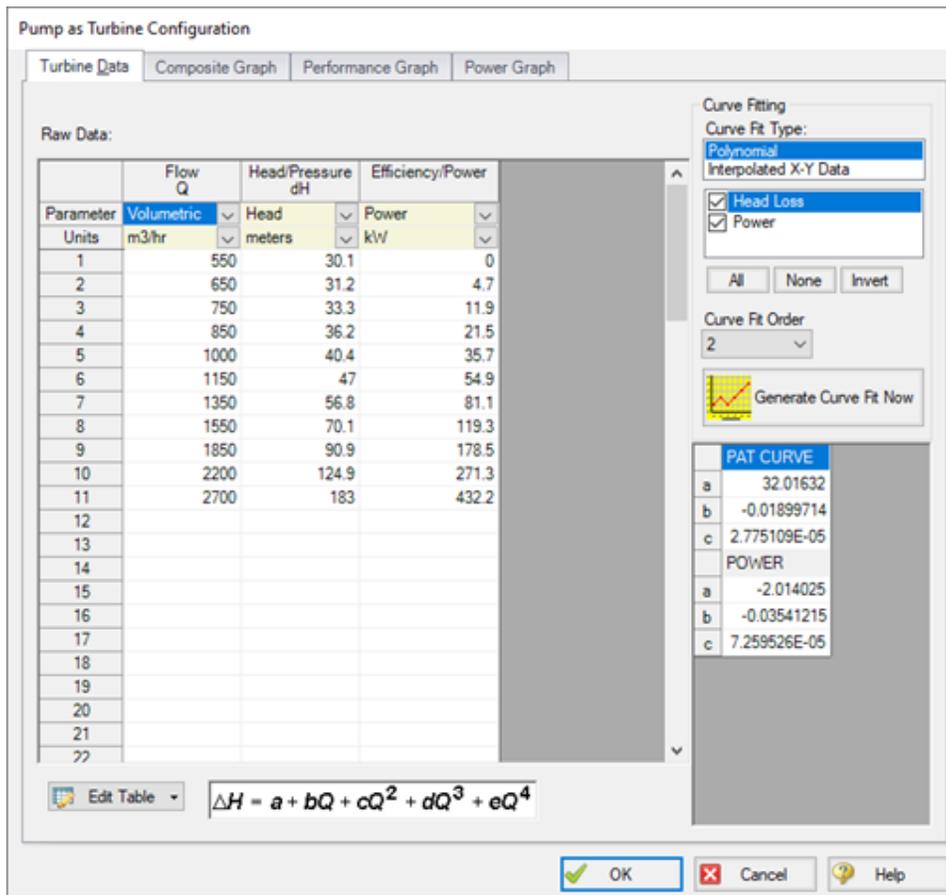


Figure 2: Pump as Turbine Configuration window

A PAT can have two curves defined:

- **Head or Pressure vs. Flow:** Required. This curve is commonly referred to as the “turbine curve” and defines the head loss across the PAT at any given flowrate.
- **Power vs. Flow:** Required. The power curve is used for two purposes for the PAT. First, it is used to determine the power generated while the pump is operating as a turbine. Second, it is used to determine if or when the pump and motor decouple.

Graphs

The following tabs display the curves that have been fit to the given data.

- **Composite Graph:** Shows the curve-fit PAT Performance and Power curves.
- **Performance Graph:** Shows the curve-fit PAT Performance curve overlaid on the raw data points. The solver will ignore the raw data points and operate on the curve fit.
- **Power Graph:** Shows the curve-fit Power curve overlaid on the raw data points. The solver will ignore the raw data points and operate on the curve fit.

PAT Operation as a Turbine

The main function of the PAT model is to allow a pump to operate as a turbine. In this mode of operation, the PAT generates power by removing energy in the form of a head loss across the pump, compared to normal centrifugal pump operation where the pump adds energy in the form of a head rise across the pump.

The turbine operating point is determined using the Turbine Performance Curve defined by the user. This curve relates head loss to flow rate similar to how the Pump Curve relates head rise to flow rate. Note that the PAT uses the turbine’s frame of reference for this data rather than the pump’s. Head loss and flow rate should be entered as positive values for the PAT. The Turbine Performance Curve should have a head loss greater than zero at zero flow and should see increasing head loss with increasing flow. An example Turbine Performance Curve is shown in Figure 1 below.

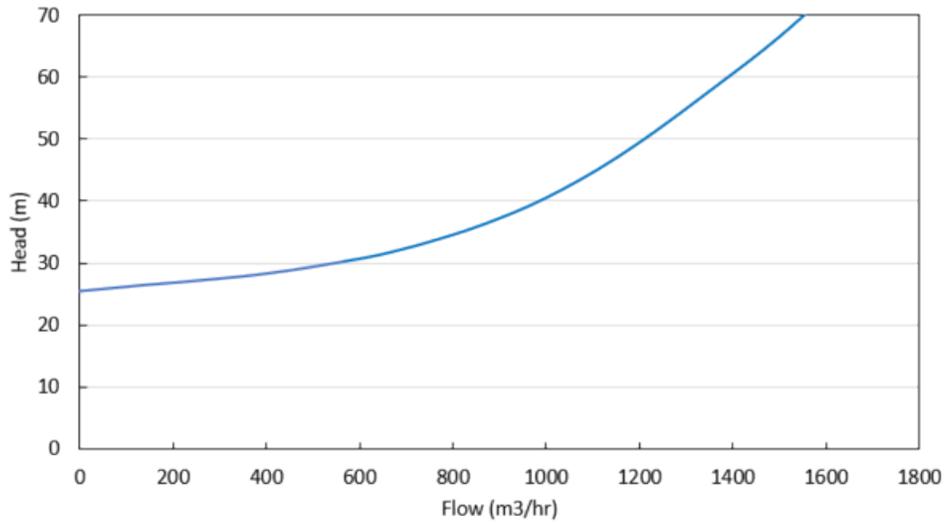


Figure 1: Example Turbine Performance Curve

The power generated by the PAT is determined by the user-defined Turbine Power Curve. This curve relates power generated to flow rate, similar to how the Pump Curve relates pump power to flow rate. Note the PAT uses the turbine’s frame of reference for this data rather than the pump’s. The Turbine Power curve should have zero power generated at some non-zero flow rate and should see increasing power output with increasing flow. An example Turbine Power Curve is shown in Figure 2 below.

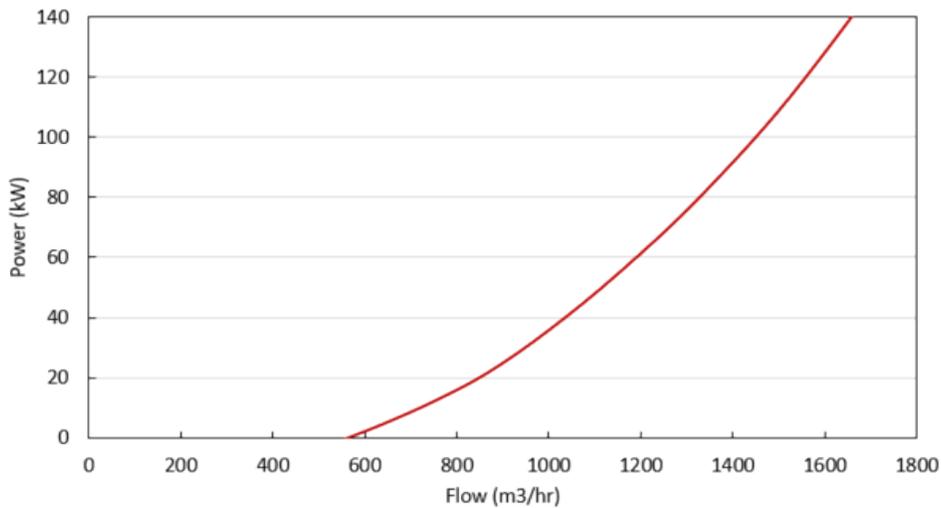


Figure 2: Example Turbine Power Curve

The flow rate at which the PAT generates zero power is called the ‘no load’ point. At this flow rate, the pump impeller is spinning with the fluid, neither adding nor removing energy from the fluid. At higher flow rates, the impeller’s speed is resisted by the motor, removing energy from the fluid and generating electricity. At lower flow rates, the impeller’s speed is resisted by the fluid, requiring energy to be added via the motor. These operating conditions are shown in Figure 3 below.

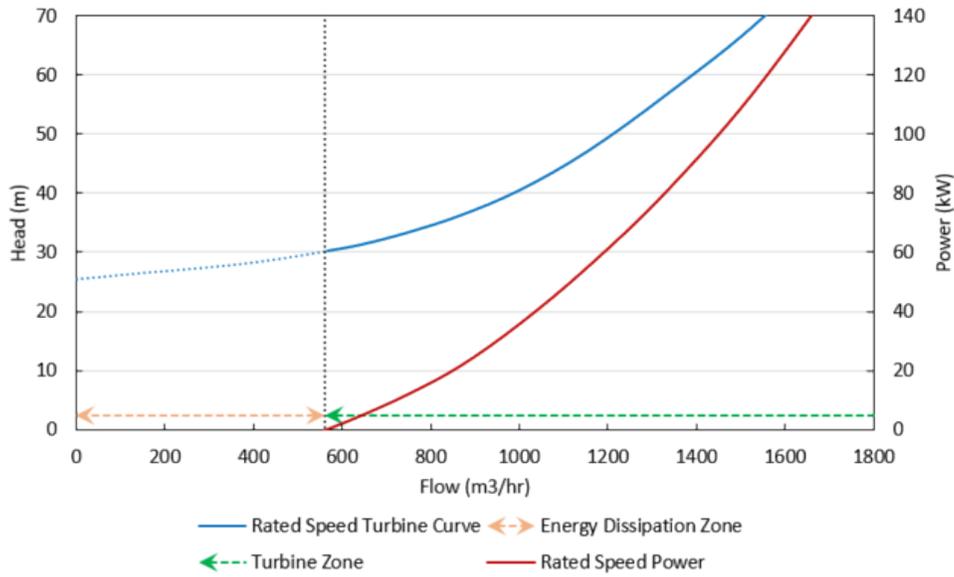


Figure 3: General turbine operating conditions

PAT Operation While Decoupled

Operation as a Turbine means the PAT is coupled to the motor, but the PAT can also operate while decoupled from the motor, meaning the pump’s impeller is spinning freely in the fluid stream. The PAT will be decoupled from the motor when it is operating below the No Load point (the flow rate with zero power generation) on the Turbine Curve, since below that flow rate the motor would be required to input power to the system.

Figure 1 below shows these operating conditions. Since the PAT is decoupled from the motor, the generator does not impart a torque on the pump impeller to counteract the torque applied by the fluid, and the pump impeller becomes free-spinning.

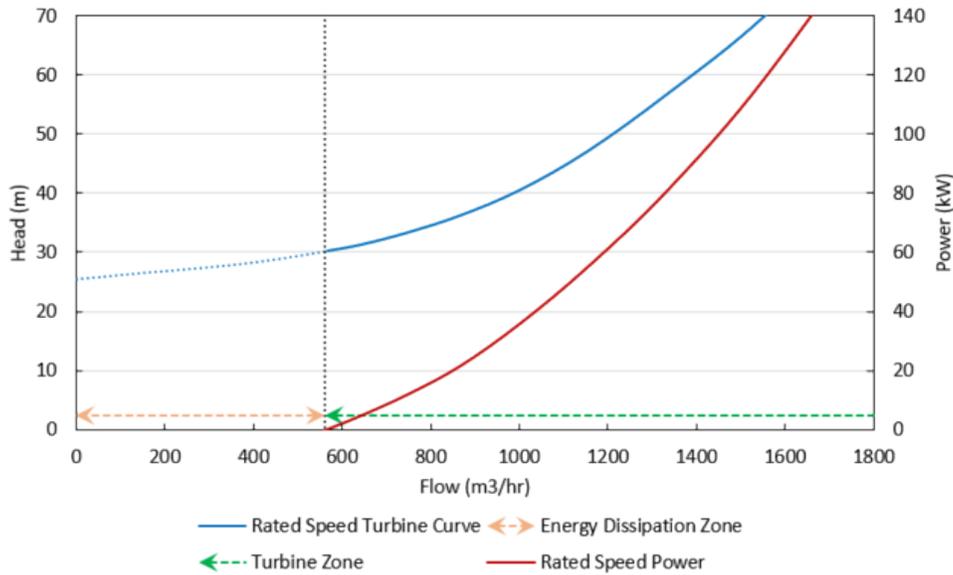


Figure 3: General turbine operating conditions

While the PAT is free-spinning, the impeller will spin at a speed dictated by the fluid such that no torque is applied on the impeller by the fluid. This speed may change based on other transients in the system and is called the runaway speed.

The pressure loss of a free-spinning PAT is determined by the Turbine Performance curve, adjusted by the Affinity Laws. The user-defined Rated Turbine Speed is considered 100% speed, and the Affinity Laws are used to adjust the curve based on speed. The Affinity Laws adjust the Turbine Performance curve in the same direction they adjust a Pump Performance curve (i.e., head loss and power increase with impeller speed).

The speed of a free-spinning impeller is determined by the system flow rate and the No Load Curve.

No Load Curve

Each runaway speed of the PAT has a different No Load point, meaning a No Load Curve can be created to describe how head loss is related to flow across the range of potential runaway speeds. Each point on the curve represents a different runaway speed and different steady state. Figure 2 below shows an example No Load curve overlaid with a Turbine Performance Curve and a Turbine Power Curve. Notice how the No Load curve intersects the Turbine Performance Curve at the point where the Turbine Power Curve has zero power.

This No Load curve can be created by plotting the head loss for multiple Turbine Performance Curves (adjusted for speed) at the flow where their corresponding Turbine Power Curves cross the zero power line. Note that AFT Fathom does not calculate, use, or require the No Load curve for its calculations. Rather, the No Load curve is merely an observable consequence of the physics the PAT follows.

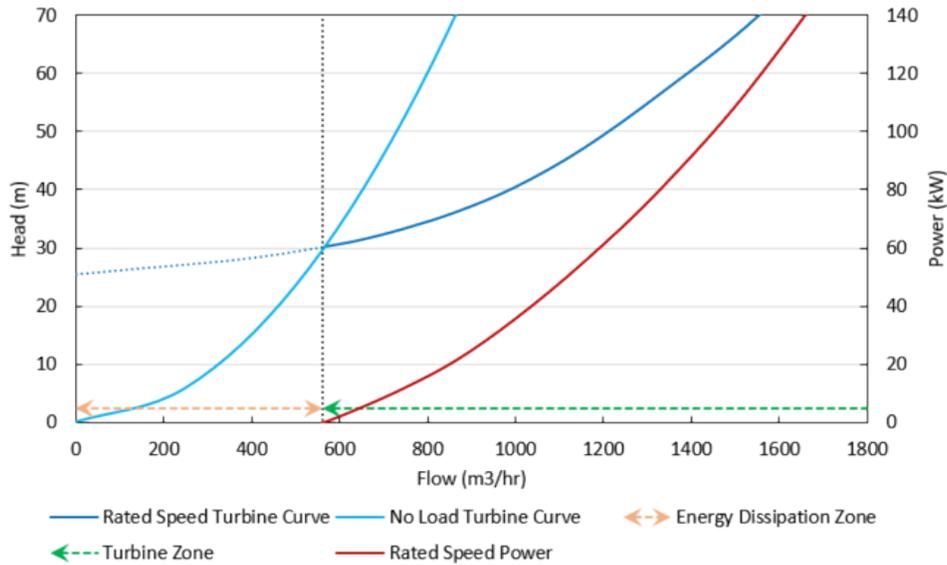


Figure 2: Example No Load Curve showing general turbine operating conditions

PAT Operation During Decoupling

The PAT transitions between the turbine and decoupled operating conditions in two situations: when manually decoupled or whenever the power generated is equal to zero. The PAT cannot recouple after becoming decoupled during a simulation.

The **first** transition from coupled to decoupled is referred to as Load Rejection and can be set as a transient event in the Transient tab of the Pump Properties window.

When this transient event is initiated, the PAT is immediately uncoupled from its motor at the onset of the transient simulation, the speed is allowed to change as dictated by the fluid, and the PAT speed is determined for each time based on the No Load Curve. The uncoupling event occurs instantaneously and the PAT will immediately begin operating on the No Load Curve at steady-state.

The **second** transition from coupled to decoupled occurs when the PAT reaches the No Load point due to other transient events in the system. This transient event is always ready to occur regardless of the transients defined for the PAT. When the flow rate through the PAT becomes low enough such that zero power is generated by the PAT, the PAT will immediately become uncoupled from the motor and become free-spinning with its speed determined by the system flow rate and the No Load Curve.

Affinity Laws

Pumps can operate at variable speeds (adjustable in the Variable Speed tab in the [Pump Properties window](#)). Affinity laws (also called homologous laws) allow an estimation of the pump performance at speeds other than the design speed.

To a somewhat lesser degree, impeller diameter changes (Impeller Modifications) follow the same laws as speed changes. Substituting impeller changes for speed changes below can be used to see the effect of an impeller change or for a small impeller trim (larger trims deviate significantly from this estimation).

Before discussing the affinity laws, it should be noted that the affinity laws are an approximation. In many cases, pump manufacturers will have performance data for a number of speeds. Manufacturer-provided data should be used if available. [Multiple Configurations](#) can be entered to save pump curves at different speeds. When data is not available for varying speeds/impeller trims, the affinity laws can be used with an understanding that an approximation is being made.

Pump curves can generally be curve fit to a polynomial:

$$\Delta H = a + b Q + c Q^2 + d Q^3 + e Q^4$$

The speed ratio is defined in terms of the design impeller speed, N_1 and the desired operating speed N_2 as:

$$s = \frac{N_1}{N_2}$$

Affinity Laws

For the following ratios, subscript 1 represents the parameter from the defined curve (assumed to be at 100% speed). Subscript 2 represents the given parameter at the desired operating speed N_2 .

- **Flow rate -**

$$\frac{Q_1}{Q_2} = s$$

- **Pump head -**

$$\frac{\Delta H_1}{\Delta H_2} = s^2$$

Note: By default NPSHr will follow the same relationship as pump head rise. The speed ratio exponent used for NPSHr can be changed on the [Configuration Data](#) tab in the Pump Configuration window.

- **Power consumption -**

$$\frac{P_1}{P_2} = s^3$$

- Efficiency -

$$\frac{\eta_1}{\eta_2} = 1$$

Related Topics

Pump.....	291
Centrifugal (Rotodynamic) Pump.....	294
Pump Configuration.....	296
Multiple Configurations.....	301
Pump Sizing.....	308
Positive Displacement Pump.....	314
Interstage Bleed/Takeoff Flow.....	309
Pump Efficiency Data.....	328
Viscosity Correction.....	313
Submerged Pump.....	326
Pump Reliability Factors.....	305

Submerged Pump

A pump junction can be defined as submerged by clicking the Submerged Pump check box in the Pump Properties window on the Pump Model tab. This is convenient to avoid the need of modeling suction piping when the suction head or pressure is a known and fixed value.

There are three options to choose from to define a submerged pump: Head (HGL), Pressure, and Surface Conditions.

Head (HGL)

With the Head (HGL) option selected, the only required value is Liquid Surface HGL. This value refers to the *liquid* surface elevation **relative to the reference elevation**, as can be seen in Figure 1. This is important when the elevation of the pump is a non-zero value. The specified elevation for the pump is then used to determine the depth of the submerged pump.

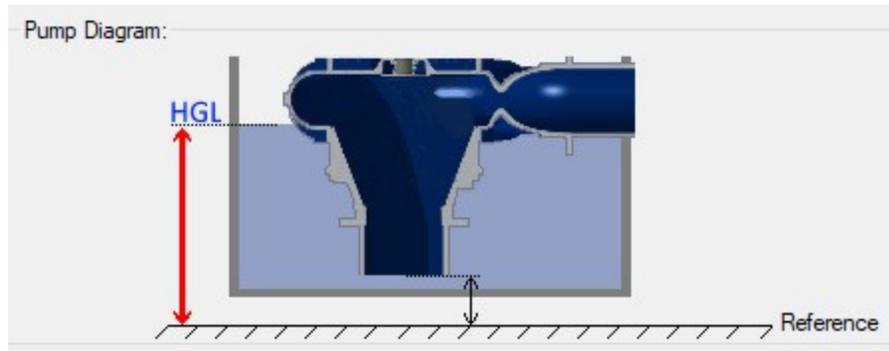


Figure 1: Diagram of Submerged Pump, Liquid Surface HGL input

Pressure

With the Pressure option selected, the only required value is Suction Pressure. This value refers to the fluid pressure at the inlet of the submerged pump.

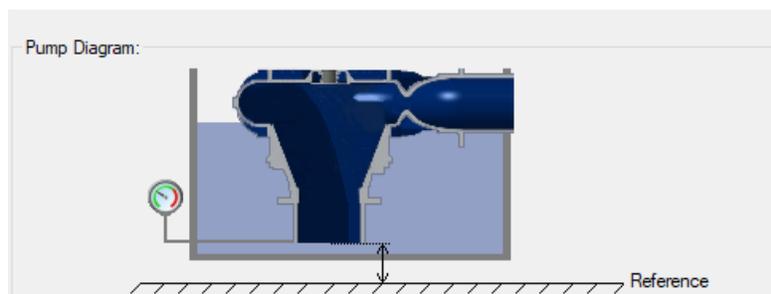


Figure 2: Diagram of Submerged Pump, Pressure input

Surface Conditions

With the Surface Conditions option selected, there are two required values: Surface Pressure and Liquid Depth. Similar to how a reservoir junction is defined, the Surface Pressure is the pressure of the fluid at the surface, which is helpful when the liquid surface is not at atmospheric pressure. The Liquid Depth is the distance between the liquid surface and the inlet of the submerged pump.

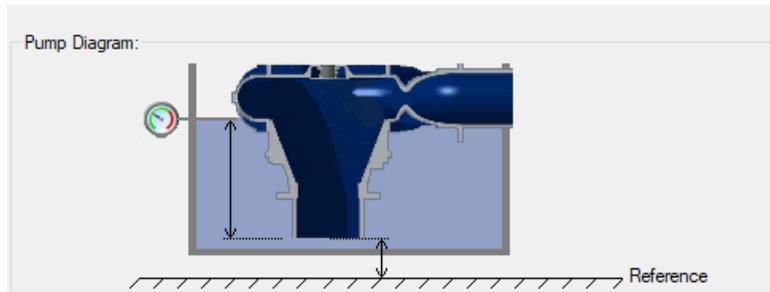


Figure 3: Diagram of a Submerged Pump, Surface Conditions input

Related Topics

Pump.....	291
Centrifugal (Rotodynamic) Pump.....	294
Pump Configuration.....	296
Multiple Configurations.....	301
Pump Sizing.....	308
Positive Displacement Pump.....	314
Interstage Bleed/Takeoff Flow.....	309
Pump Efficiency Data.....	328
Viscosity Correction.....	313
Affinity Laws.....	324
Pump Reliability Factors.....	305

Pump Efficiency Data

Best Efficiency Point (BEP)

By defining an efficiency or power curve, the Best Efficiency Point of the pump can be found. Optionally, the calculated value can be overridden with a user-defined value in the [Pump Configuration](#) window. It is generally desirable to operate pumps as close to the BEP as possible. The Pump Summary tab in the Output can display the BEP, and the % of BEP which indicates how close the operating point is to the BEP.

Power and Efficiency Calculations

Various stages of power or efficiency data relating to a pump assembly can be added to a pump junction. The most basic value that can be accounted for is the pump efficiency or power curve. This curve, entered in the [Pump Configuration](#) window, is used to determine the power/efficiency being delivered to the pump itself - the Shaft Power.

The Solver will determine the total dynamic head (ΔH) and flow (Q) through the pump via iteration for the unique system. Then, the **Ideal Power** (also called hydrodynamic power) can be directly calculated using the equation below:

$$P_{Ideal} = Q\rho g\Delta H$$

where ρ is the fluid density at the pump and g is the gravitational acceleration. When a pump efficiency or power curve is entered on the Pump Configuration window, the power supplied to the pump, or **Shaft Power**, can be calculated:

$$P_{Shaft} = \frac{P_{Ideal}}{\eta_{Pump}}$$

where the pump efficiency comes from the pump curve at the given flow. On a pump's optional tab, further efficiency information can be defined:

	Motor Power	Motor Eff. (%)	VFD Power	VFD Eff. (%)
1				
2				
3				

Power Units: hp

Edit Table

Figure 1: Additional Efficiency Data

Entries in this table must come in pairs - both Power and Efficiency values must be present. For example, if Motor Power is entered, the Motor Efficiency column must also be entered. You can enter only motor data, only VFD data, or both.

Motor Power in this instance refers to the *output* power of the motor, which is equal to the Shaft Power calculated previously. This is because the motor driver is mechanically coupled to the shaft and impeller. Because the Shaft Power is already known, it can be looked up inside this table using linear interpolation, and the corresponding Motor Efficiency is used to calculate the **Motor Input Power**:

$$P_{Motor\ Input} = \frac{P_{Shaft}}{\eta_{Motor}}$$

VFD Power is used in a similar fashion. The VFD Power in Figure 1 refers to the *output* power of the VFD, which is identical to the Motor Input Power (but instead of being mechanically coupled, this is represented by an electrical connection). Because Motor Input Power was calculated, VFD Efficiency can be found from the table using linear interpolation, and **VFD Input Power** can be calculated:

$$P_{VFD\ Input} = \frac{P_{Motor\ Input}}{\eta_{VFD}}$$

VFD Input Power is equivalent to the **Overall Power**, viewable in the Pump Summary along with the other above values. Whenever an efficiency value is left undefined, it is assumed to be 1 (perfectly efficient).

$$\eta_{overall} = \eta_{VFD}\eta_{Motor}\eta_{Pump} \quad P_{Overall} = \frac{P_{Ideal}}{\eta_{overall}}$$

Related Topics

Pump.....	291
Centrifugal (Rotodynamic) Pump.....	294
Pump Configuration.....	296
Multiple Configurations.....	301
Pump Sizing.....	308
Positive Displacement Pump.....	314
Interstage Bleed/Takeoff Flow.....	309
Viscosity Correction.....	313
Submerged Pump.....	326
Affinity Laws.....	324
Pump Reliability Factors.....	305
Junction X %BEP (Y%) is above/below the POR range.....	779
Poor Pump Curve Fits.....	800

Related Examples

Hot Water System.....	950
-----------------------	-----

Relief Valve

The Relief Valve junction allows users to model a valve that opens at a specified pressure in the system. Depending on how the relief valve operates, that pressure could either be the pressure at the relief valve inlet or the pressure at some remote point in the system. The differences between those valves are discussed below.

The Relief Valve Properties window follows the first of the [two basic Properties Window formats](#), displaying the connected pipes in a fixed format. The Relief Valve junction adopts a flow direction from the connecting pipes.

There are three types of relief valves available:

- **Internal Relief Valve:** contains two connecting pipes and relieves into the downstream pipe. This type of relief valve acts like a regular valve that is closed until the specified pressure is reached.
- **Exit Valve:** contains one connecting pipe and relieves to an external ambient pressure when the specified pressure is reached.
- **Inline Exit:** contains two connecting pipes and relieves to an external ambient pressure when a specified pressure is reached. This type of relief valve acts as a lossless connection when closed and allows flow between the connecting pipes.

Opening and Closing Profiles

The Relief Valve junction has several methods by which the valve can open and close. These options are:

- **Instant:** the relief valve opens fully or closes fully from one time step to the next as the opening or closing conditions are met.
- **Pressure:** the valve passively opens or closes according to a C_v vs. Pressure profile as the opening or closing conditions are met. The C_v vs. Pressure profiles are based on the Valve Setpoints and the data entered in the Loss Model tab. Maximum C_v vs. Time rate limits can be applied.
- **Never:** the valve never closes. This method cannot be applied to the valve opening.

Balanced vs Unbalanced Valves

Relief valves can either be hydraulically balanced (constant backpressure) or non-hydraulically balanced. This characteristic of the valve is determined by how the dome of the relief valve is pressurized.

For an unbalanced valve, the dome of the valve is pressurized via the fluid downstream of the valve. Since the downstream pressure can change during the transient, the valve position is based on a pressure differential between the upstream and downstream pipes rather than based solely on the upstream pressure.

For the balanced valve, the dome of the valve is pressurized to a constant pressure, often by attaching some type of bellows device. With this system, the valve position is based only on the actual upstream pressure.

Valve Setpoints

The valve setpoints determine when the relief valve will open and when it will be closed. These setpoints can be defined using either pressure or head.

There are four possible valve setpoints the user needs to define:

- **Exit Pressure:** the ambient pressure downstream of the relief valve for the Exit or Inline Exit valve types. This pressure will only be used to determine when the valve opens and closes when the relief valve is unbalanced.

- **Overpressure:** the upstream pressure at which the valve is fully open. This input is only needed when the valve opens or closes based on pressure.
- **Set Pressure:** the upstream pressure at which the valve opens. This input must be specified for all valve profiles and is replaced by the High Pressure Setpoints when Remote Sensing is enabled.
- **Blowdown Pressure:** the upstream pressure at which the valve is fully closed. All options for valve opening and closing require this input, with the exception of the Never profile. If the relief valve closes at the same pressure where it opens, the Set Pressure and the Blowdown Pressure should be set to the same value.

Remote Sensing

The Relief Valve junction gives users the ability to model Remote Sensing Relief valves. This option is available anytime the valve operates as a pressure/pressure valve.

A Remote Sensing Relief Valve is a relief valve that opens based on the pressure at a specified location in the system other than the upstream side of the valve. In a transient system, this design allows the relief valve to be more responsive to surge events by opening the valve before the pressure surge arrives at the valve itself. In a steady state system, this design allows the valve to open at some remote pressure while the degree to which the relief valve is open is determined by the pressure at the valve.

Loss Model

The Relief Valve loss information is entered on the Loss Model tab. The available options change depending on how the valve opening and closing are defined. For all options, the Base Area for Loss Model can be specified, with the upstream pipe being used by default.

The following loss model options are available depending on which profiles are selected:

- **Instant Profiles**
 - **C_v (Constant):** the relief valve instantly changes from closed to a fixed C_v .
 - **K_v (Constant):** the relief valve instantly changes from closed to a fixed K_v .
 - **C_dA (Constant):** the relief valve instantly changes from closed to a fixed C_dA . This C_dA is defined as the product of an Orifice Effective Area (A) and a Discharge Coefficient (C_d).
 - The Orifice Effective Area can come from an API 526 letter designation or it can be User Specified.
 - The Discharge Coefficient must always be specified by the user. The value is typically specified by the manufacturer.
 - **K-Factor:** the relief valve instantly changes from closed to a fixed K-Factor.
 - **Resistance curve:** the relief valve instantly changes from closed to a resistance curve.
- **Pressure Profiles**
 - **C_v (Variable):** the relief valve moves between the closed and open states with the C_v varying between 0 and the Fully Open C_v value.

- **K_V (Variable):** the relief valve moves between the closed and open states with the K_V varying between 0 and the Fully Open K_V value.

The Variable Data option for the Relief Valve Loss Model allows users to model the valve losses based on where the valve is between fully open and fully closed. The Linear based on setpoints option determines those losses solely based on the setpoints the user has specified. If the user knows enough about the valve to describe a detailed loss profile, they can select the Non-linear option. The first data point must have a loss value of 0 at the Blowdown pressure. The final data point must have a loss value equal to the Fully Open loss value, at the Overpressure. The entered data is interpolated during the transient simulation.

Restricted Flow Area

An optional input in the Relief Valve Properties Window is the Restricted Flow Area. This parameter describes the effective area restriction in the valve. This is done for the purpose of assessing cavitation. In most cases, information on the size of the restricted area must be obtained from the manufacturer or from test data.

Special Conditions

The relief valve has two Special Conditions:

- **Failed Open:** the valve is forced to remain open regardless of pressure conditions.
- **Ignore Relief Valve:** the valve is forced to remain closed regardless of pressure conditions.

For more information see Special Conditions.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Kv and Cv Loss Model.....	263
Unique solution not possible - Relief valve cycles.....	783

Reservoir

The Reservoir junction type allows you to connect up to twenty-five pipes. At least one connecting pipe is required. The Reservoir junction is convenient for specifying a fixed pressure in your system.

A Reservoir junction applies a defined pressure and temperature (if using [heat transfer](#)) in the model at the junction location. When solving a pipe flow system, a Reservoir causes the rest of the system to distribute the flow in a manner consistent with the defined pressure.

The Reservoir Properties Window follows the second of the [two basic window formats for junctions](#).

The Liquid Surface Elevation of the Reservoir must be defined - this value will be used along with the Pipe depth to determine the hydrostatic pressure contribution at the pipe inlet.

Surface Pressure is required, and fluid temperature is required when heat transfer is modeled.

Specifying Pipe Connectivity

If there is a single pipe connected to the junction, its elevation can be directly specified on the Reservoir Model tab. The elevation can be specified either as an absolute Elevation, or as a depth below the liquid surface. This means there are two ways to define the same pipe elevation.

If multiple pipes are connected, they can each have a unique elevation. This is displayed on the Pipe Depth and Loss Coefficients tab. In this table, all elevations must be specified as either depth or elevation, as is specified by the radio button selection below the table. Additionally, losses such as re-entrant losses can be added to the pipe connections in this table. Loss factors can optionally be specified for flow into and flow out of the pipe.

Pipes Connected Above Liquid Surface

A pipe elevation can be specified higher than that of the liquid surface. Pipes that empty into the Reservoir above the liquid surface are assumed to have liquid free fall to the liquid surface. AFT Fathom applies the proper boundary condition for above the liquid surface.

If the pipe is above the liquid surface, the only physically realistic condition is for the fluid to be flowing from the pipe into the reservoir. However, AFT Fathom will assume the fluid flowing into the pipe is the same as the reservoir fluid in order to solve the system. If this occurs a [warning](#) will be shown in the Output.

Infinite and Finite Reservoirs with the XTS Module

With the standard AFT Fathom, all reservoirs are treated as infinite. With the [XTS Module](#), reservoirs can be infinite or finite.

- **Infinite reservoir**
 - If modeling an infinite reservoir, the user can vary the surface level or surface pressure with time. Note that the liquid level is not calculated in this case, but is input by the user.
- **Finite tanks**
 - Finite tank reservoirs can be open or closed. Finite reservoirs require specification of tank geometry. Typically, the liquid level and surface pressure are defined initially, and then calculated over time based on the tank geometry.
 - Open tanks do not vary the surface pressure unless the user does so through the Transient tab. Closed tanks vary the surface pressure in accordance with gas thermodynamic laws and a user specified polytropic constant.

- For closed tanks, users can enter maximum and minimum pressures. These represent pressure control systems on the tank such as relief valves. Liquid overflow lines can be modeled with normal pipes. In such cases use of valves with event transients can be used.
- **Known parameters initially**
 - In some cases, the initial liquid level or pressure may not be known, and it is desired to have this calculated based on a mass balance in the system. This can be selected using the check box options provided.

If an XTS model is run, and the pipe is no longer submerged with flow into the pipe, AFT Fathom will stop the flow in that pipe on subsequent time steps because there is no liquid to supply it.

A potential confusion exists when pipe connection data is entered as depth below the liquid surface, and then the liquid surface changes during a transient. This is not an issue in that all depth data is converted internally to absolute elevation at the beginning of the run so that liquid level changes will be handled properly.

Tip: Since transient models allow the liquid level to change, and the user may desire to run different scenarios with different initial liquid levels, it is better to specify reservoir pipe connections based on elevation and not depth.

Energy Balance

The energy balance feature is displayed only when heat transfer is being modeled. For an open system, most users desire that the temperature remains fixed, like the pressure does. However, for closed, recirculating systems, you will usually want the temperature to vary. The temperature can be allowed to vary at a junction by checking the Balance Energy option. See [Open vs. Closed Systems - Balancing Energy](#) for detailed discussion.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Finite and Infinite Reservoirs.....	601
Pipe X elevation of Y is below Reservoir Z tank bottom elevation.....	788
Tank Junction X began overflowing at Y seconds.....	782
Tank Junction X completely drained at Y seconds.....	782
Flow is into system at Reservoir Junction X.....	777

Screen

The Screen junction type requires two connecting pipes. This junction type allows you to model the irrecoverable loss that occurs through a screen, mesh, or filter flow restriction. It also allows you to specify loss factors as a function of a flow parameter.

The Screen Properties Window follows the first of the [two basic Properties Window formats](#), displaying the connecting pipes in a fixed format. The Screen junction does not have an explicit flow direction; it adopts a flow direction from the connecting pipes.

AFT Fathom provides four standard screen types for your use:

- Sharp-edged perforated plate
- Round-edged perforated plate
- Circular wire screen
- Silk thread screen

The loss factors for these screens are taken from handbook values. In addition to the standard screens that AFT Fathom provides, you may enter your own custom screen loss factor that is consistent with the application.

When you select Resistance Curve in the Loss Model section on the Loss Model tab, the Screen Properties window makes additional features available. Using these features, you can input pressure drops that vary with flow. To enter this information, you can specify polynomial constants, [fit a curve](#) to available data, or use interpolated x-y data.

When a variable loss is specified, AFT Fathom modifies the loss factor in the Solver to agree with the solution. You can choose any of the optional flow and pressure parameters provided, and you can specify the most convenient units.

An optional input in the Screen Properties Window is the Restricted Flow Area. This parameter describes the effective area restriction in the screen. This is done for the purpose of assessing cavitation. In most cases, information on the size of the restricted area must be obtained from handbook or test data.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Screen Loss Model.....	336

Screen Loss Model

The screen loss factors have much in common with orifice loss factors. In fact, the **sharp-edged perforated plate** and **round-edged perforated plate** screen losses are the same as for orifices with similar area ratios and equal upstream and downstream areas. Refer to the [orifice loss factor](#) equations for these relationships. The relationships are good for screen area Reynolds numbers above 100,000.

The **circular metal wire screen**, shown in Figure 1, is given by the following equation for screen area Reynolds numbers above 1000 ([Idelchik 2007](#), 598):

$$K_{up} = 1.3 \left(1 - \frac{A_{screen}}{A_{up}} \right) + \left(\frac{A_{up}}{A_{screen}} - 1 \right)^2$$

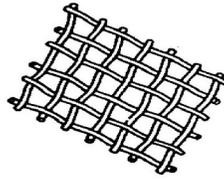


Figure 1: Circular metal wire screen

The **silk thread screen**, shown in Figure 2, is given by the following equation for screen area Reynolds numbers above 500 (Idelchik 2007, 577):

$$K_{up} = 2.1 \left(1 - \frac{A_{screen}}{A_{up}} \right) + \left(\frac{A_{up}}{A_{screen}} - 1 \right)^2$$

For other screen configurations, see Chapter 8 of Idelchik's [Handbook of Hydraulic Resistance \(2007\)](#).

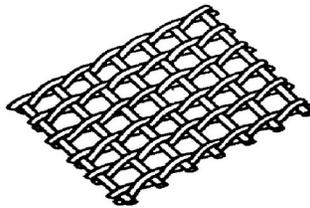


Figure 2: Silk thread screen

Related Topics

K Factor Loss Model.....	261
Bend Loss Model.....	274
Area Change Loss Model.....	269
Orifice Loss Model.....	288
Tee/Wye Loss Model.....	342
Valve Modeling Closures with K Factors.....	352
Screen.....	335

Spray Discharge

The Spray Discharge junction must have at least one connecting pipe, and it allows up to four. This junction offers a flexible way of modeling a flow exiting the system through a nozzle or spray device.

The Spray Discharge Properties window follows the second of the two basic [Properties Window formats](#). A table displays the connecting pipe information. The pipe table grows in size to accommodate up to four connecting pipes.

The exit pressure to which the Spray Discharge junction discharges must be entered. If a [heat transfer](#) fluid model is used, then exit temperature must also be entered.

Two geometries can be modeled. The first is a normal, one hole spray, where the discharge flow area of the hole is entered. The second is a sparger which, in principle, works in the same way as a spray. For the sparger, the flow area of a single hole and the number of holes are entered. This assumes that the holes are hydraulically close in proximity and are the same area.

For each connecting pipe, the pipe table on the Pipe Losses tab lists the reference flow direction and two loss factors. The first loss factor is for physical flow out of the pipe and into the junction, the second is for physical flow into the pipe and out of the junction. The loss factors can be specified independently or left as zero. To edit within the pipe table, click in the appropriate column and row.

The distance of each connecting pipe (if any difference exists) from the specified junction elevation can be entered in the pipe table.

Loss Model

The [discharge coefficient](#) for the junction is entered under the Loss Model tab in the box near the bottom of the window. The physical area of the exit flow can also be entered under the discharge coefficient input box. These two parameters are used to calculate the discharge flow rate, which depends on the difference between the internal pressure in the system and the specified exit pressure.

The [K Fire Sprinkler](#) model uses the K value as defined by the fire sprinkler industry. Whereas the standard K loss value is dimensionless, the K Fire Sprinkler has units associated with it. You can adjust these units according to your data.

If you select to model the spray discharge pressure losses with a Resistance Curve, the Spray Discharge Properties window makes additional features available. Using these features, you can input the spray pressure drop that varies with flow. To enter these factors, you can specify polynomial constants, [fit a curve](#) to available data, or use interpolated x-y data.

Special Conditions

Spray Discharge junctions have the following [Special Conditions](#) on the Optional tab:

- **None** - Flow is allowed to enter/exit the system at the junction as determined by the system hydraulics and external pressure.
- **Closed** - Flow is not allowed to enter/exit the system at the junction, but flow can still go in/out of the connected pipes as determined by the system hydraulics.
- **No Inflow** - Flow is not allowed to enter the system from the surroundings, but is allowed to exit the system as determined by the system hydraulics and external pressure.

Flow Net vs Through

The [Output Control](#) allows reporting of Mass/Volumetric "Flow Rate Net At Junction" or "Flow Rate Through Junction". Depending on the number of connected pipes and flow directions determined by the

Solver, the Flow Rate Through Junction may report values of N/A. This is because there may not be a unique flow path "through" the junction.

- **Net At Junction** - Generally this parameter is measuring the flow entering/exiting the system. It is calculated by summing the flow into the junction and subtracting the sum of flow out of the junction. A positive value indicates flow exiting the system, and a negative value indicates flow entering the system.
- **Through Junction** - Generally this parameter is measuring the flow through a unique path in the junction if one is available. A unique flow path could include a scenario where the Special Condition is set to Closed, but flow enters from one upstream pipe, passes through the junction (lossless), and exits into one downstream pipe. Another example of a unique flow path is if the Spray has only one connected pipe.

Note: Pipes with inverted [Reference Positive Flow Directions](#) and negative flowrates can cause confusion when interpreting results with the Flow Rate Net At Junction and Flow Rate Through Junction parameters. Care should be taken when observing the signs of the parameters.

Transient

([XTS Module](#) Only) On the Transient tab the CdA or K data can be varied with time to simulate the opening or closing of the junction. For more information on transient data, including event transients, see the [Junction Transient Data](#) topic.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Spray Discharge Fire Sprinklers.....	339
Spray Discharge CD (Discharge Coefficient).....	340

Related Examples

Spray Discharge System.....	874
NFPA Firewater System.....	934
Spray Discharge System - GSC.....	1019

Spray Discharge Fire Sprinklers

The definition of K value for fire sprinklers is:

$$K_{sprinkler} = \frac{Q}{\sqrt{\Delta P}}$$

where Q and ΔP can be in the units of your choice. The preference in the USA is gal/min for flowrate and psid for pressure. In Europe it is frequently liter/min for flow rate and bars for pressure. Note that $K_{sprinkler}$ **is not dimensionless**, but has units associated with it. Also, note that sprinkler vendor values for $K_{sprinkler}$ implicitly assume that water is the fluid, so there is a built-in density of 62.3 lb_m/ft³ or 1,000 kg/m³. If this sprinkler is used for something other than water, the $K_{sprinkler}$ value will need to be adjusted.

Also note that if the spray is into the atmosphere, as is typical, the ΔP is the same thing as the gage pressure. That is why the previous equation is stated by some vendors as just P in the denominator, rather than ΔP . When stated as just P , the pressure is gage.

Finally, note that the Exit Pressure, as defined by AFT Fathom, is different than the concept of Discharge Pressure, an often used parameter in the Fire Protection industry. AFT software defines exit pressure as the pressure of the ambient surroundings, which in an unpressurised location at sea-level, is 1 atm.

Note: The fire sprinkler K is not the same as the dimensionless K value as given by [Equation 1](#) in the Loss Model Summary topic and used extensively throughout AFT Fathom.

The $K_{sprinkler}$ is related to the discharge coefficient. It can be shown that the relationship is given by

$$C_D A = \sqrt{\frac{\rho_{water}}{2}} K_{sprinkler}$$

Related Topics

Spray Discharge	337
Spray Discharge CD (Discharge Coefficient)	340

Related Examples

NFPA Firewater System	934
-----------------------------	-----

Spray Discharge CD (Discharge Coefficient)

The pressure drop across a spray discharge which uses a discharge coefficient is discussed here. The mass flow rate through a restriction is given as follows:

$$\dot{m} = \rho C_D A V$$

The pressure drop across a discharging restriction is given as:

$$\Delta P = \frac{1}{2} \rho V^2$$

Combining the two yields:

$$\Delta P = \frac{1}{2} \rho \left(\frac{\dot{m}}{C_D A \rho} \right)^2$$

or

$$\Delta P = \frac{1}{2 \rho (C_D A)^2} \dot{m}^2$$

This equation is used for spray discharge junctions in AFT Fathom, with the ΔP being the difference between stagnation pressure at the spray discharge inlet and the exit pressure specified for the junction.

Related Topics

Spray Discharge	337
Spray Discharge Fire Sprinklers	339

Related Examples

Spray Discharge System	874
Spray Discharge System - GSC	1019

Tee/Wye

The Tee/Wye junction type requires three connecting pipes.

Loss Model

There are two Loss Models available for Tee/Wye junctions: Simple and Detailed. If you choose the Simple type of Tee/Wye, no loss factors will be calculated. This is equivalent to a branch with no loss factors entered for the connecting pipes. In some systems, the losses associated with a Tee/Wye are small compared to the overall loss in the system. To avoid unnecessary calculations, you should choose the Simple type of Tee/Wye unless you need the actual losses.

If the Detailed loss model is chosen, additional features will become available to define the arrangement of the pipes connected to the tee/wye, as well as information on the tee/wye geometry. By default, Fathom will automatically define the straight and branch run pipes based off of the arrangement in the Workspace. If Fathom cannot determine an arrangement, the user will be required to define the pipe arrangement in the Tee/Wye Properties window. The arrangement defined in this window will affect the calculated loss factors.

The Detailed type of Tee/Wye will calculate the loss factors as they depend on the flow split as well as the geometry. Because of the dependence on the flow split, loss factors usually cannot be calculated before performing an analysis. AFT Fathom incorporates four models to calculate tee and wye loss factors during the actual solution. These calculations account for the developed flow pattern, requiring additional

iteration. See [Tee/Wye Loss Model](#) and [Tee/Wye Junction Complexity](#) for a detailed discussion of Detailed Tee Losses.

If you have specific loss factors that you want to use for a Tee/Wye junction, you should use a [Branch](#) junction instead and enter the appropriate K factors on the Optional tab.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Tee/Wye Loss Model.....	342
Tee/Wye Junction Complexity.....	803

Tee/Wye Loss Model

The Simple loss model assumes there are no losses across the tee/wye. The Detailed loss model uses various correlations and data to calculate the loss coefficients across the different flow paths of the tee/wye. Generally, the correlations depend on the flow split, ratio of pipe cross-sectional areas, and angle of the connecting pipes.

- **Sharp Tee** - This option represents tees with sharp corners ($r/d=0$) that have equal areas in the header and an equal or smaller area in the branch ($A_c = A_a = A_b$).
- **Rounded Tee** - This option represents tees with rounded or smooth corners ($r/d=0.1$) that have equal areas in the header and an equal or smaller area in the branch ($A_c = A_a = A_b$).
- **Area Change Tee** - This option represents tees with sharp corners ($r/d=0$) which have an area sum ($A_a + A_c = A_b$).
- **Symmetrical Wye** - This option represents symmetrical wyes with sharp corners ($r/d=0$) which have an area sum ($A_a + A_b = A_c$).

If the cross-sectional area requirements are violated, then the software will still provide results because it accounts for different basis areas and effectively "corrects" the loss coefficient by using velocity ratios. However, it is important to note that in these cases the correlations are still being used outside of their original ranges, so there is inherently some additional uncertainty.

Nomenclature

The correlations and data used by the Detailed loss model come from two different sources: Idelchik's Handbook of Hydraulic Resistance and Miller's Internal Flow Systems (see References below). Each source has their own naming conventions and nomenclature. Furthermore, both nomenclatures assume that the user knows the flow directions ahead of time. For these reasons, AFT has selected a more general nomenclature to describe the names of each pipe in the tee/wye. AFT's nomenclature also accounts for the fact that the user may not know the flow directions ahead of time, and might not be able to properly specify the tee using Idelchik's or Miller's conventions. Since most of the correlations come from Idelchik, and many of our users are familiar with the Idelchik text, we will first describe some of the important nomenclature here. Then, the AFT nomenclature will be depicted below. For reference, Idelchik uses

several unusual parameter names; the greek letter Zeta represents the loss coefficient (K factor), "F" represents the cross-sectional area of the pipe, and "w" represents the velocity.

Figure 1 depicts the Idelchik nomenclature, but using "A" for area instead of "F". The pipes are labeled as combined flow (c), straight-through flow (st), and side branch (s). The combined flow pipe always carries the total flow, and depends on if the tee has a diverging or converging flow pattern (flow patterns discussed further in the next section). Loss coefficient subscripts refer to a basis area and location in the tee. For example, $K_{c,s}$ represents the K factor for the side branch (s) referenced to the combined (c) pipe flow area and velocity. An issue with applying the Idelchik convention to a general software is that sometimes the flow directions are not known ahead of time, or may change based on multiple operational configurations, transient events, etc.

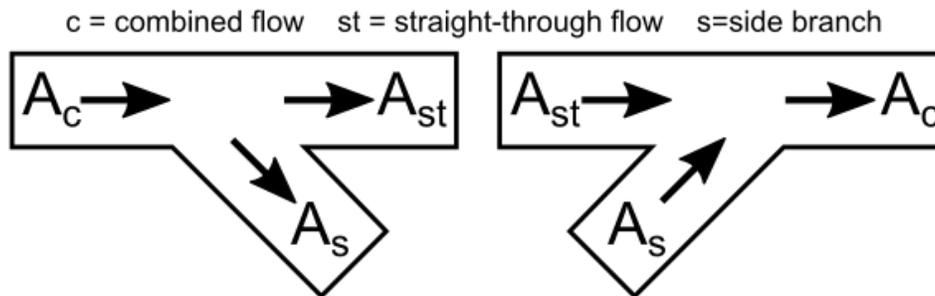


Figure 1: Idelchik's nomenclature for diverging (left) and converging (right) flow through a Tee/Wye junction

Figure 2 depicts the AFT nomenclature. The pipes are labeled as a, b, and c. The c pipe is always the branching pipe (equivalent to the "s" pipe in Idelchik). This convention allows AFT to cover all flow pattern possibilities discussed below and listed in Table 1 without specifying exactly which pipe is the combined flow pipe.

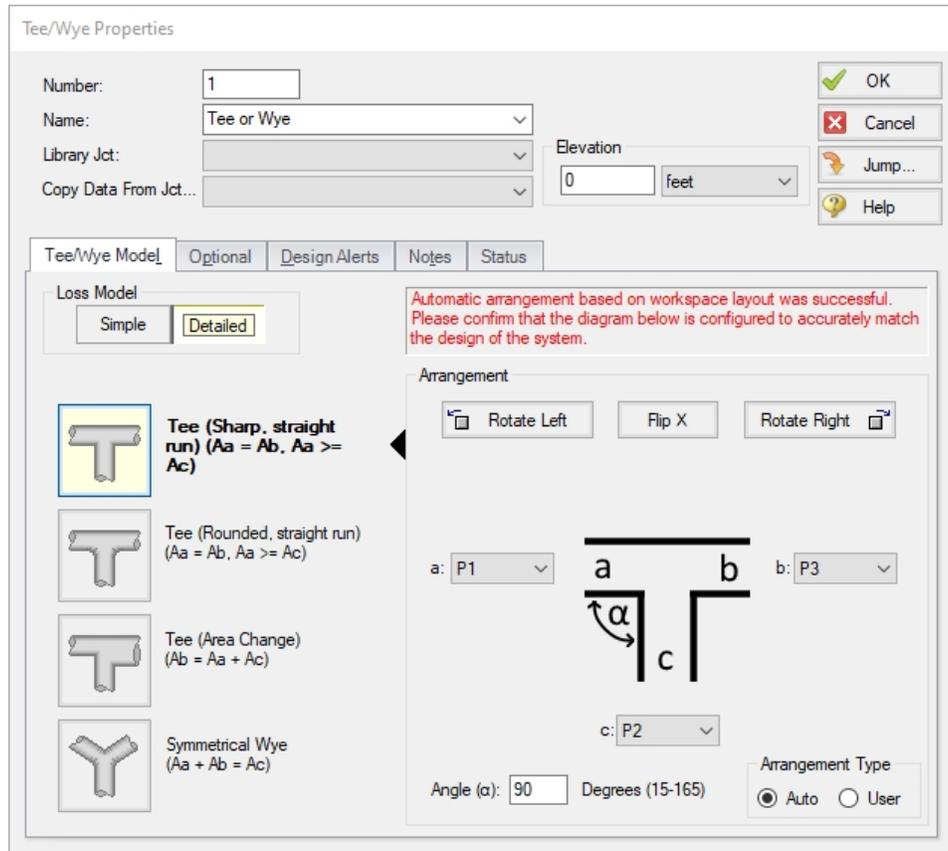


Figure 2: AFT's nomenclature in the Tee/Wye Properties window where the c pipe is always the branch pipe

Flow Patterns & Correlation Selection

There are four general types of flow patterns in tees which depend on the directions of the flow streams:

1. **Diverging From Header** - flow comes from a single upstream pipe in the header, and a portion of flow diverges into the branch pipe while the remaining portion continues in the straight run.
2. **Converging Into Header** - flow comes from two upstream pipes, one of which is the header and other is the branch pipe, and the two streams converge into the downstream portion of the header.
3. **Dividing From Branch** - flow from the branch pipe divides in opposite directions in the header.
4. **Combining Into Branch** - flow comes from opposite directions in the header and combines into the branch pipe.

The Solver will automatically determine which flow pattern is applicable based on the actual flow directions in the model. The Diverging and Converging patterns use equations or tabulated data with interpolation that come from Idelchik. The Dividing and Combining patterns use tabulated data with interpolation that comes from Miller.

Angle

If the tee angle is 90 degrees, it is not important which pipe you call the *a* pipe and which the *b* pipe. The distinction between *a* and *b* takes on more importance for non-90 degree pipes. AFT will always define the angle from the *a* pipe to the *c* pipe. The user should define the angle according to this definition and be consistent with their physical layout of the pipes. If the actual flow directions in the final solution are different, then the Solver will automatically handle this in the backend.

Some Idelchik correlations are for specific, discrete angles. If the user-entered angle is between these values, then the software will interpolate. Other Idelchik correlations provide equations with the angle as an input, and are valid for a specified angle range. If the angle falls outside of the range, then it will be treated as either the maximum or minimum angle.

The data referenced from Miller was only for 90 degree tees, and therefore, the angle has no effect and is always assumed to be equal to 90 degrees for the Dividing From Branch and Combining Into Branch flow patterns.

Convergence Issues

Sometimes intermediate iterations of the Solver cause a different flow pattern to be selected, and sometimes the system may experience instability as the Solver gets stuck switching back and forth between two different flow patterns. Other times, having many Detailed tees in a pipe network can cause numerical complexity due to the effects of the tees coupling together in such a way that the Solver cannot converge on all of the flow splits. The tee loss coefficients are dependent on the flow splits, but the flow splits are dependent on the other losses in the system. Therefore, if a pipe network contains many tees using the Detailed loss model, when one tee's loss coefficient is updated from an intermediate Solver iteration, the entire network needs to be re-calculated to determine how the flow splits changed. Sometimes this process is not able to terminate within the Tolerances specified by the model.

See the [Tee/Wye Junction Complexity](#) help topic for more information and suggestions. In general, if the losses from the tees are small in comparison to the total system losses, or other components that contribute significant pressure loss, then it may be acceptable to assume the tee is lossless and use the Simple loss model.

Negative Loss Coefficients

Under the right conditions it is possible to see a negative loss coefficient for one of the flow paths in the tee. This is physically realistic and is supported in literature by both Idelchik and Miller. In summary, a negative loss coefficient represents an energy/pressure gain for said flow stream of the tee and is achieved by having a portion of kinetic energy transferred to it from the other flow stream. The other flow stream sees an increased loss coefficient (higher than it would have been otherwise), and the net result is still an irrecoverable energy loss across the entire tee control volume. Some specific details are discussed below.

- **Converging Tee Side Branch:** Turbulent mixing and momentum exchange takes place between the particles which favors equalization of the velocity field within the pipe which carries the merged

flow. In other words, there is a stream with higher energy (velocity/kinetic energy), and a stream with lower energy, and when they merge together they level out to some equilibrium that is somewhere between the high and low energy in the combined flow pipe. The stream which had the higher velocity effectively transfers part of its kinetic energy to the stream with the lower velocity.

- Evidence from Idelchik 3rd Ed 1994: Diagrams 7-1 to 7-9, the tables and graphs show negative $K_{c,s}$ values.
- Evidence from Miller 2nd Ed 1990: Figure 13.10 pg 311, the chart plots lines of constant K factor, some of which have negative values.
- **Diverging Tee Straight Run:** At certain flow rate ratios (Q_s/Q_c), a portion of the slowly moving boundary layer adjacent to the wall passes into the side branch while transferring part of the momentum into the straight passage. The increase in energy in the straight passage is associated with a negative K_{st} , while also accompanied by an increased positive value of K_s such that the whole process is a net irrecoverable pressure loss.
 - Evidence from Idelchik 3rd Ed 1994: Diagrams 7-1 to 7-9, the tables and graphs show negative $K_{c,st}$ values.
 - Evidence from Miller 2nd Ed 1990: Figure 13.23 pg 319, the curve dips below zero for a range of flow ratios between 0.1 and 0.4.

Larger Branching Flows Down a Header

There is an unintuitive, but realistic, situation that can occur due to modeling tees with the Detailed loss model. In some cases, as flow goes down a header pipe past a series of tees, there may be larger flow rates out of the final branching pipes compared to the initial branching pipes (see Figure 3). This is a realistic phenomenon that is caused by the "turning resistance" being larger than the "straight-through resistance"; or in other words, the loss factor for the flow to turn and go out of the branching pipe is larger than the loss factor for the flow to continue straight through the tee. Depending on the boundary conditions and losses in the remaining downstream parts of the network, this may influence the flow to take the path of least resistance and go straight through the tee as opposed to turning and flowing out of the branching pipe.

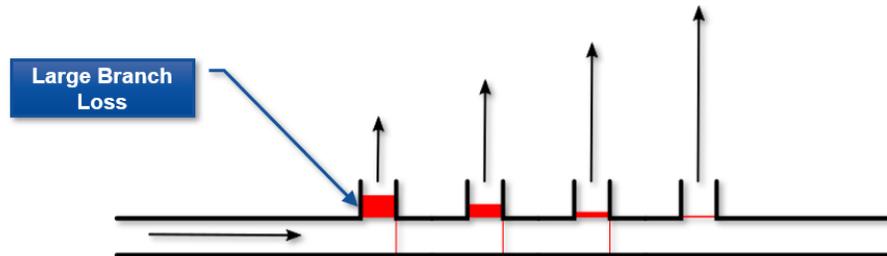


Figure 3: Illustration of branching flow increasing down the length of a header with a series of Detailed tees

By comparison, a tee modeled as lossless (Simple loss model) has no turning resistance or straight-through resistance. Stagnation pressure decreases continuously down the length of the header due to friction. As a result, the branching flow rates will decrease correspondingly (see Figure 4). The results in Figure 4 with decreasing flow down the header are also possible with the Detailed loss model, and

whether you see results more similar to Figure 3 or Figure 4 depend on the pipe network as a whole, including the other losses in the system and the boundary conditions.

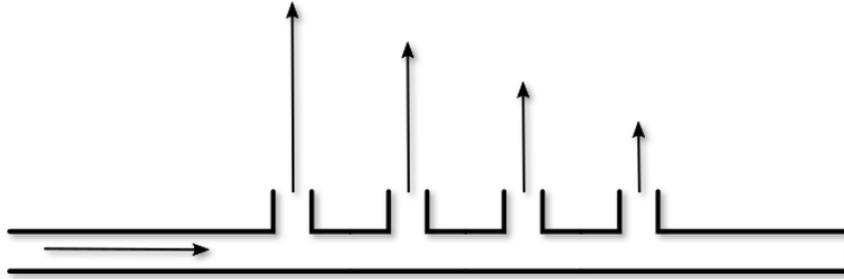


Figure 4: Illustration of branching flow decreasing down the length of a header (possible with Simple or Detailed loss models)

References

The correlations used by the Detailed loss model are taken mostly from Chapter 7 of Idelchik's Handbook of Hydraulic Resistance ([Idelchik 2007](#)), with some additional data taken from Miller's Internal Flow Systems ([Miller 1990](#)). Table 1 lists which references are used based on the user's selection in the Tee/Wye Properties window and which flow pattern is applicable as determined by the Solver.

Junctions

Table 1: References for Tee/Wye junction correlations

Geometry	Type	Direction	Reference	Page	Diagram / Figure
Constant Area, Sharp (r/D = 0)	Diverging	Side Branch	Idelchik 2007	523	7-18
	Diverging	Straight Run	Idelchik 2007	525	7-20 No. 1
	Converging	Side Branch	Idelchik 2007	501-504	7-1 to 7-4
	Converging	Straight Run	Idelchik 2007	501-504	7-1 to 7-4
	Dividing	From Branch	Miller 1990	323*	13.27 (r/d=0)
	Combining	Into Branch	Miller 1990	315*	13.16 (r/d=0)
Constant Area, Rounded (r/D = 0.1)	Diverging	Side Branch	Idelchik 2007	531-533	7-23 No. 1
	Diverging	Straight Run	Idelchik 2007	533 -> 526-527	7-23 -> 7-20 No. 1
	Converging	Side Branch	Idelchik 2007	511-515	7-11 to 7-13
	Converging	Straight Run	Idelchik 2007	511-515	7-11 to 7-13
	Dividing	From Branch	Miller 1990	323*	13.27 (r/d-d=0.1)
	Combining	Into Branch	Miller 1990	315*	13.27 (r/d-d=0.1)
Area Sum, Sharp (r/D = 0)	Diverging	Side Branch	Idelchik 2007	524	7-19
	Diverging	Straight Run	Idelchik 2007	526-527	7-20 No. 2
	Converging	Side Branch	Idelchik 2007	505-509	7-5 to 7-9
	Converging	Straight Run	Idelchik 2007	505-509	7-5 to 7-9
	Dividing	From Branch	Miller 1990	323*	13.27 (r/d=0)
	Combining	Into Branch	Miller 1990	315*	13.16 (r/d=0)
Wye	Diverging	-	Idelchik 2007	545 -> 524	7-30 -> 7-19

Geometry	Type	Direction	Reference	Page	Diagram / Figure
	Converging	-	Idelchik 2007	545	7-30
<p>* Estimates have been made on available data. -> Means the first page number points the reader to a different Diagram.</p>					

Related Topics

K Factor Loss Model.....	261
Bend Loss Model.....	274
Area Change Loss Model.....	269
Orifice Loss Model.....	288
Screen Loss Model.....	336
Valve Modeling Closures with K Factors.....	352
Tee/Wye.....	341
Tee/Wye Junction Complexity.....	803

Three Way Valve

The Three Way Valve junction type always connects to three pipes. This junction type allows you to model the irrecoverable losses that occur through a valve with multiple inlets or outlets.

The Three Way Valve Properties Window does not follow either of the [two basic Properties Window formats](#), but instead shows the connecting pipes in drop-down lists. This allows specification of which pipe is the combined flow pipe and which two are the split flow pipes.

All losses in a Three Way Valve are based on C_v (or K_v). The C_v data is entered for various percent open positions, and the user ultimately specifies the actual percent open to be used in the model. The resulting C_v values for the two flow directions are calculated and displayed. A graph of the open percentage, C_v , and flow area data can be displayed by clicking the Show Graph... button located below the Open Percentage Data table.

Special Conditions

When the specified percent open value input into the Actual Percent Open box corresponds to a C_v of one or both flow directions to be zero, a Special Condition is applied to the corresponding flow direction (s), causing the valve to close in any direction in which the C_v is zero.

Transient

([XTS Module](#) Only) On the Transient tab, a transient open percent can be entered to simulate the position changes of the valve over time. For more information on transient data, including event transients, see [Junction Transient Data](#).

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Kv and Cv Loss Model.....	263

Related Examples

Beginner - Cost Calculation.....	908
Controlled Heat Exchanger Temperature - GSC.....	1027

Valve

The Valve junction type requires two connecting pipes, unless you specify it as an exit valve, in which case only one connecting pipe is allowed. This junction type allows you to model the irrecoverable loss that occurs through a flow control component. The Valve Properties window follows the first of the two basic [Properties window formats](#), displaying the connecting pipes in a fixed format. A flow direction through the junction is adopted from the defined directions of the connecting pipes.

Loss Source

In order to completely define a valve junction, you must specify the valve loss source by clicking on the appropriate radio button under Loss Source:

- **User Specified** - This is the best option if manufacturer data is known. Manually input the loss information using one of the Loss Models listed in the next section.
- **From Characteristic** - This option is useful if no information about the valve is known aside from the type, and an estimate needs to be made. Automatically calculate C_V based on the selection of a common valve geometry type and a Full Open C_V value. With the exception of the User Defined characteristic from the dropdown, the window will automatically populate an Open Percent vs C_V data table by scaling the shape of the characteristic curve to fit the provided Full Open C_V . Then, choose an Open Percentage under the Loss Parameters area to automatically calculate a C_V . The data for the characteristic shapes comes from [Val-Matic 2009](#).
- **Handbook** - Uses standard valve loss models from [Idelchik, Miller, and Crane](#).

Loss Model

Depending on the selection of the Loss Source, various loss models will be displayed:

- [Flow Coefficient](#) - C_V for English units or K_V for Metric units
- [K Factor](#)
- [Resistance Curve](#)

- [Equivalent Length](#)
- [3-K \(Darby\)](#)

A special **Reduced Port** configuration is available when using the Handbook loss source, and requires the selection of a valve type and several geometric inputs. Once specified, a K Factor from the Handbook source will be calculated.

Exit Valves

Valves that are used as exit flow control mechanisms are specified as Exit Valves. These valves require an associated ambient pressure definition (also called a back pressure). Exit Valves can only be connected to a single upstream pipe. To specify an Exit Valve, select the Exit Valve checkbox, and input the required Exit Pressure. If a [heat transfer](#) fluid model is used, the Exit Temperature must also be specified to define the thermal boundary condition.

Restricted Area & Optional Losses

The Restricted Area is optional input available for the User Specified and Handbook loss sources which describes the effective area restriction in the valve for the purpose of assessing cavitation. By default, Fathom will check for cavitation in pipes since an area is specified as part of the required pipe properties. Cavitation in a junction can only be checked if a Restricted Area is provided. In most cases, information on the size of the restricted area must be obtained from the manufacturer or test data.

Note: Fathom does not model cavitation but can provide a warning if the static pressure is below the vapor pressure. If a User Defined fluid is used, the optional input parameter for Vapor Pressure must be defined in the Fluid Properties. Fathom only models single phase flow, and assumes all pipes are fluid-full as stated in the [Engineering Assumptions](#).

The [Basis Area](#) is available for the K Factor, Resistance Curve, Equivalent Length, 3-K (Darby), and Reduced Port loss models and describes the area from which loss factors are referenced from.

Open Percentage Data

Optional data may be entered for special valve characteristics. Specifically, the Open Percentage vs C_v and Flow Area can be specified. This data does not affect the Solver's flow solution; however, engineers frequently desire to know the valve's open percentage during operation to ensure it meets design requirements. The Valve Summary in the Output window always displays the valve C_v , and if data is specified for open percentage and flow area it will also display open percentage and flow area at the operating point. A graph of the open percentage, C_v , and flow area data can be displayed by selecting the Characteristic Graph tab below the table.

The Characteristic Source parameters can be used to create an Open Percent vs C_v curve that follows the shape of several common valve types including Ball, Butterfly, [Equal Percentage](#), Globe, Linear, Plug, and Quick Opening.

Transient (XTS Module Only)

The Transient tab is only visible if the [XTS Module](#) is active, and a transient C_v or K Factor vs time table can be entered to simulate the opening or closing of the valve. If the optional open percentage table is entered, it can also be used to specify an open percent vs time transient. For a closed valve, the K Factor is infinite. To account for this, a K Factor of -1 (negative one) can be used in the transient data table. For more information on transient data, including event transients, see the [Junction Transient Data](#) topic.

Special Conditions

The [Special Condition](#) on the Optional tab allows the valve to be specified as Closed, which is useful to turn off a valve while retaining the input data for future use.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Kv and Cv Loss Model.....	263
Valve Modeling Closures with K Factors.....	352

Valve Modeling Closures with K Factors

A closed valve would have an infinite K-factor. Since infinity is not a possible input, AFT Fathom uses the convention of $K = -1$ (negative one) to represent a closed valve.

This input is only valid when defining a valve transient while using the [Fathom XTS Module](#). To represent a valve that is closed in steady state, use the valve Special Conditions in the Optional tab.

Related Topics

K Factor Loss Model.....	261
Bend Loss Model.....	274
Area Change Loss Model.....	269
Orifice Loss Model.....	288
Screen Loss Model.....	336
Tee/Wye Loss Model.....	342
Valve.....	350

Venturi

The Venturi junction type requires two connecting pipes. This junction type allows you to model the irrecoverable loss that occurs through a venturi area change, which is frequently used in conjunction with

pressure taps in flow measurement devices. It also allows you to specify loss factors as a function of a flow parameter.

The Venturi Properties window follows the first of the two basic [Properties Window formats](#), displaying the connecting pipes in a fixed format. The Venturi junction does not have an explicit flow direction, but adopts a flow direction from the connecting pipes.

Loss Models

AFT Fathom does not provide any standard venturi loss models. Venturi flow characteristics vary from vendor to vendor, so it is difficult to find generally applicable models. You must enter your own custom venturi loss factor that is consistent with the application. This information should be obtained from the manufacturer or from test data.

For convenience, you can specify constant loss characteristics of a venturi as a discharge coefficient (C_d) or a loss factor (K).

If you would like to model the pressure loss through the Venturi as varying with the flow rate, select the radio button next to Resistance Curve under the Loss Model tab of the Venturi Properties window. Once Resistance Curve is selected, additional features appear that allow you to input pressure drops versus flow data. To enter this data, you can specify polynomial constants, [fit a curve](#) to available data, or use interpolated x-y data.

When a variable loss is specified, AFT Fathom modifies the loss factor in the Solver to agree with the solution. You can choose any of the optional flow and pressure parameters provided, along with the most convenient units.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261

Volume Balance

Note: The Volume Balance junction must be used with great caution. It is only intended to model specific situations and when used incorrectly will give incorrect results. When modeling mixing fluids with [varying fluid properties](#), the Output may show a [Caution message about the density change](#). This does not necessarily mean that a Volume Balance should be used at the mentioned location. Refer to Figure 1 below for an example of when the Volume Balance junction can be used appropriately.

The Volume Balance junction type is internal to the piping system and requires two connecting pipes.

The Volume Balance Properties window follows the first of the [two basic Properties window formats](#), displaying the connecting pipes in a fixed format. The flow direction through the junction is determined by the defined directions of the connecting pipes.

The Volume Balance junction is useful for modeling pipelines with a moving interface between fluids of different density. Such a scenario is, in reality, a transient application, because the mass in the system is

not steady with time. Fathom is designed to balance mass at all branches, except for the Volume Balance junction. For the Volume Balance, the volume is balanced between the upstream and downstream pipes. This is consistent with the fact that an equal velocity exists at the interface, resulting in a volume balance at the interface.

Volume Balance junctions should be used in conjunction with the Variable Fluids feature, without heat transfer, but with variable density (see [Heat Transfer/Variable Fluids panel](#) in [Analysis Setup](#)).

The usage of a Volume Balance junction is illustrated with the Slurries With Variable Fluid Properties - SSL example ([English](#), [Metric](#)). Figure 1 below provides a summary of the four snapshots in time that occur in the Slurries with Variable Fluid Properties example. In reality, injecting slurry into a pipeline where there are two separate fluids with an interface between is a transient situation.

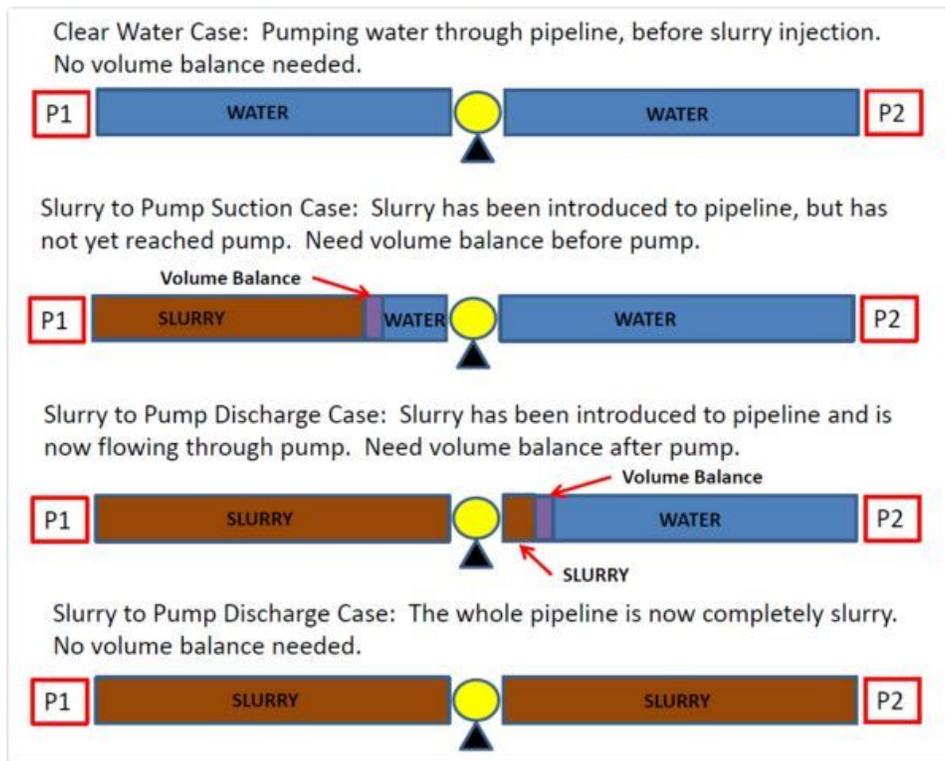


Figure 1: Four snapshots in time and when to use the volume balance junction is shown for slurries and variable fluid properties systems

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261
Junction X has a density change across it.....	786

Related Examples

Slurries with Variable Fluid Properties - SSL1102

Weir

Note: The Weir junction must be used with great caution. A weir inherently is for open-channel flow. However, Fathom always assumes flow is in a pipe, and that the pipe is liquid full. The loss models for the Weir junction will give a good estimate of losses associated with a weir in a system, but does not truly model open-channel flow.

The Weir junction type requires one or two connecting pipes. The Weir Properties window follows the first of the [two basic Properties window](#) formats, displaying the connecting pipes in a fixed format. A flow direction through the junction is adopted from the defined directions of the connecting pipes.

When using the Weir junction, keep in mind that AFT Fathom only models fluid flow in liquid-full pipes. Use of the Weir junction will not allow AFT Fathom to model open-channel flow.

Weir Types

There are three weir configurations available. The following input data is required for each.

- **Rectangular Sharp Crested -**
 - Surface Pressure
 - Notch Elevation (Z_n)
 - Notch Width (L)
 - Channel Width (B)
 - Number of contractions

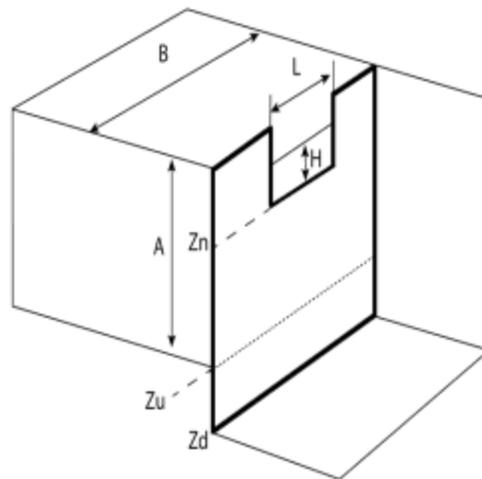


Figure 1: Rectangular Sharp Crested Weir Model

- **V-Notch Sharp Crested -**
 - Surface Pressure
 - Notch Elevation (Z_n)
 - Notch Angle (θ)
 - Channel Width (B)

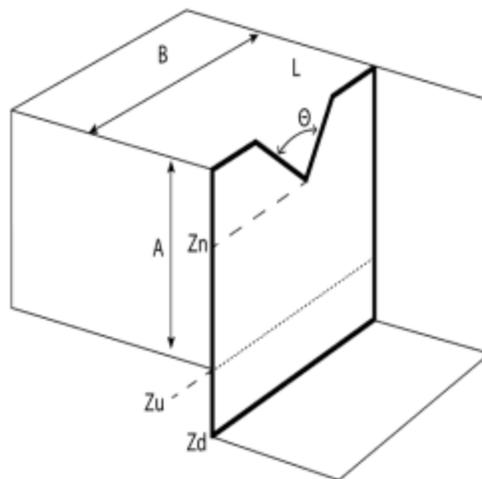


Figure 2: V-Notch Sharp Crested Weir Model

- **Cipolletti, where the vertical angle is a 1:4 ratio -**
 - Surface Pressure
 - Notch Elevation (Z_n)

- Notch Width (L)
- Channel Width (B)

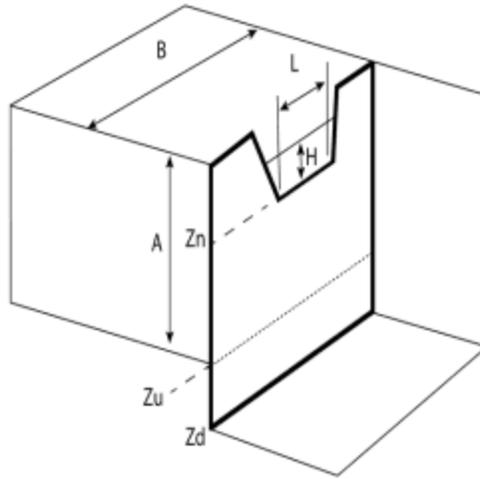


Figure 3: Cipolletti (1:4 ratio) Weir Model

Notch elevation (Z_n)

The absolute elevation of the bottom of the notch using the same reference frame as the pipe elevation.

Elevation of the Upstream and Downstream Areas (Z_u and Z_d)

The absolute elevation of the bottom of the the upstream (Z_u) tank area and downstream (Z_d) tank area using the same reference frame as the pipe elevation.

Number of Contractions

For a Rectangular Sharp Crested weir the notch area can be contracted from the sides of the weir. This can be from one side (1 contraction) or both (2 contractions). If the weir notch extends the full length of the channel then there are no contractions.

Upstream and Downstream Area Geometry

If the Cross-Sectional Area is specified as Constant, the weir area is assumed to be a vertical tank with constant cross-sectional area. If the cross-sectional area changes with height, this can be entered by selecting the variable option and entering the cross-sectional variation in the table.

The Tank Cross-Sectional Area is used for the purpose of determining the liquid volume. It will determine how high the liquid will rise or drop, since a given volume of liquid will flow into the weir during a time step.

Related Topics

Junctions.....	248
General Junction Information.....	249
Junction Loss Models.....	261

Interface and User Options

This section focuses on the how the items within the various menus and tabs that comprise AFT Fathom can be used to customize the user's experience within the software. Topics discussed include the options present in each of the AFT Fathom drop-down menus, the tools available in the Primary Window tabs, and descriptions of the customizable parameters in the User Options menu.

Contents

Menus and Toolbars	359
Printing	370
Primary Window Overview	372
Status Bar	373
Quick Access Panel	374
Model Status Light	383
Solution Progress Window	383
Workspace Window	384
Model Data Window	408
Output Window	413
Graph Results Window	426
User Options Overview	446

Menus and Toolbars

Many of the actions required to set up an AFT Fathom model can be completed from the various menus located at the top of the AFT Fathom window. Additionally, these menus can provide helpful features such as saving, importing and exporting files, and accessing various help documents such as the product help files and examples. These menus include the...

- [File menu](#)
- [Edit menu](#)
- [View menu](#)

- [Analysis menu](#)
- [Tools menu](#)
- [Library menu](#)
- [Arrange menu](#)
- [Window menu](#)
- [Help menu](#)

This topic also gives a high level overview of [Toolbars](#), including the Main Toolbar as well as each Primary Windows toolbar.

File Menu

- **New** - Opens new model.
- **Open** - Calls Open file common dialog window so a pre-existing model can be selected for opening.
- **Save** - Saves input model using current model file name.
- **Save As** - Opens Save As common dialog so you can specify a file name to save model.
- **Save Graph As** - (Graph Results Only) - Opens Save Graph As common dialog so user can specify a file name to save graph as a graphics image file.
- **Save Graph Data As** - (Graph Results Only) - Opens Save Graph Data As common dialog so user can specify a file name to save graph x-y data to text file.
- **Import Graph Data** - (Graph Results Only) - Allows for data as .txt and .dat to be imported for creating graphs.
- **Import Excel Change Data** - (Workspace and Model Data Only) - Imports data from Excel file which modifies certain input data.
- **Excel Export Manager** - Opens the [Excel Export Manager](#).
- **Export Model Data** - (Model Data Only) - Opens formatting window and Save As dialog where you can specify data from the Model Data window to be saved to a delimited file appropriate for spreadsheet use. Available in Model Data window.
 - Exporting the Model Data as a Preformatted Report will generate an *.inp file, which is not to be confused with [EPANET files](#) that have the same file extension. These files can be useful for performing text comparison to find differences which is similar to the [Scenario Comparison Tool](#).
- **Export Output** - (Output Only) - Opens formatting window and Save As dialog where you can specify data from the Output window to be saved to a delimited file appropriate for spreadsheet use.
- **Merge** - (Workspace Only) - Opens a Merge dialog where you can specify another model file to merge into the current model.
- **Print Preview** - Opens Print Preview which allows you to specify special formatting for printed graphics and text, as well as preview printed content before sending it to the printer.
- **Start Batch Run** - The Start Batch Run button on the File Menu opens a Batch Run dialog where you can select a different scenarios or models and initiates a process whereby Fathom runs the multiple scenarios/models automatically.

- **Load Background Picture** - (Workspace Only) - Loads a background picture into the Workspace. Formats supported include bitmap, windows metafile, enhanced metafile, JPEG, GIF and RLE. Available in Workspace window.
- **Import Piping Layout From**
 - **Piping Component File (.pcf)** - [Import a Piping Component File](#)
 - **CAESAR II Neutral File (.cii)** - [Import a CAESAR II Neutral File](#)
 - **GIS Shapefile (.shp)** - Allows for [Importing GIS Shapefile](#) (geospatial maps) as .shp file.
 - **EPANET File (.inp)** - [Import an EPANET File](#)
- **Export Piping Layout To**
 - **EPANET File (.inp)** - [Export an EPANET File](#)
- **Exit** - Exits AFT Fathom.

Related Topics

Workspace Window.....	384
Model Data Window.....	408
Output Window.....	413
Graph Results Window.....	426
Menus and Toolbars.....	359

Edit Menu

- **Undo** - (Workspace Only) - Undoes the previous workspace manipulation action. Fathom will keep track of a historical list of changes, and users can revert the workspace back to recent states.
- **Redo** - (Workspace Only) - Reverts the workspace following Undo actions. Fathom will keep track of undo actions, and users can redo any recent changes that have been undone.
- **Cut** - (Workspace Only) - Cuts the selected pipes and junctions on the Workspace and places them on the clipboard.
- **Copy** - (Workspace, Model Data, and Graph Results Only) - Copies the selected contents to the clipboard.
- **Copy X-Y Data** - (Graph Results Only) - Copies numerical data from graph to the Windows clipboard. Available in the [Graph Results window](#).
- **Copy Graphics** - (Workspace Only) - Copies selected [Workspace](#) content to the Windows clipboard. Formats supported include bitmap, windows metafile, and enhanced metafile.
- **Paste** - (Workspace Only) - Pastes clipboard pipes and junction contents (from previous cut or copy operation) into the Workspace model. New paste preview mode allows the pasted pipes and junctions to be moved and specifically placed where desired and will indicate interference with other junction by changing color. A green box indicates that there is no interference and a red box indicates that the junctions will be overlapping and are not in a proper location. Available in the Workspace window.
- **Delete** - (Workspace Only) - Deletes the selected pipes and junctions on the Workspace.
- **Duplicate** - (Workspace Only) - Duplicates the selected pipes and junctions on the Workspace.

- **Duplicate Special** - (Workspace Only) - Opens [Duplicate Special](#) dialog window where you can specify the increment by which all pipes and junction numbers will be increased when the model is duplicated, the creation of a new group from the duplicated objects, and whether to hide the duplicated objects in the output.
- **Select All** - (Workspace Only) - Selects all pipes and junctions on the Workspace.
- **Select Last Selection** - (Workspace Only) - The most recent pipes and/or junctions selected may be reselected.
- **Select Special** - (Workspace Only) - Opens [Select Special](#) window where group of pipes and junctions can be built up.
- **Select Flow Path** - (Workspace Only) - Selects the sequence of pipes and/or junctions in the [flow path](#) connected to the selected pipe(s) on the Workspace.
- **Select Shortest Path** - (Workspace Only) - The shortest flow path between two junctions can be found and selected by Fathom. To enable this option, two junctions must be selected in the Workspace which Fathom will find a path between.
- **Customize Icon** - (Workspace Only) - Allows user to change junction icon color, orientation, and icon image.
- **Special Conditions** - (Workspace Only) - Sets or clears Special Conditions for selected pipes and junctions on the Workspace.
- **Groups** - (Workspace Only) - Allows you to create and manage groups of pipes and junctions.
- **Renumber** - (Workspace Only) - Allows renumbering pipes and junctions on the Workspace using one of the [Renumbering](#) methods.
- **Join Selected Pipes/Jcts** - (Workspace Only) - A continuous series of connected pipes and junctions can be combined into a single pipe object to simplify the Workspace. See [Joining and Splitting Pipes](#).
- **Split Pipe at Intermediate Elevations** - (Workspace Only) - A pipe with intermediate elevations can be split into separate pipes at the intermediate elevation points. See [Joining and Splitting Pipes](#).
- **Global Pipe Edit** - (Workspace and Model Data Only) - Opens Global Pipe Edit window where editing of multiple pipes can be performed.
- **Global Junction Edit** - (Workspace and Model Data Only) - Opens Global Junction Edit window where editing of multiple junctions can be performed.
- **Transfer Results to Initial Guesses** - (Output Only) - Copies solution results into initial guesses for pipes and junctions.
- **Make Selection Same As Parent** - (Workspace Only) - Replaces all input data for the selected pipes and/or junctions with data from those objects in the current scenario's parent.
- **Clear Initial Guesses** - Clears all initial guesses in pipes and junctions.

Related Topics

Workspace Window.....	384
Model Data Window.....	408
Output Window.....	413
Graph Results Window.....	426

Menus and Toolbars.....	359
-------------------------	-----

View Menu

- **Last View** - (Workspace Only) - Switches to the most recent Workspace view. The view is changed by scrolling, zooming, or using the [Workspace Find](#). The Last View in the View Menu will take you back to the previous view of the Workspace. You can toggle between two views by continuously clicking Last View.
- **Select Objects** - (Workspace Only) - Changes cursor to selection tool to select junctions, pipes, etc.
- **Overview Map** - (Workspace Only) - Opens overview map in the [Quick Access Panel](#).
- **Workspace Pan** - (Workspace Only) - Changes cursor to pan tool to move around workspace.
- **Show Magnifier** - (Workspace Only) - Turns on the [Magnifier tool](#) in the Workspace.
- **Show Object Status** - (Workspace Only) - Toggles [Show Object Status](#), which checks status of all pipes and junctions in model.
- **List Undefined Objects** - (Workspace Only) - Opens [Undefined Objects window](#) which displays undefined pipes and junctions and their properties.
- **Extended Model Check** - Performs an exhaustive check of model consistency and completeness.
- **Highlight in Pipe and Jct Windows** - (Workspace Only) - Toggles [Highlight feature](#) for pipe and junction Properties Windows.
- **Color Common Size Groups** - (Workspace with [ANS Module](#) Only) - Colors the pipes assigned to the selected Common Size Group(s) in the Workspace.
- **Color Design Requirements** - (Workspace with [ANS Module](#) Only) - Colors the pipes assigned to the selected Design Requirement in the Workspace.
- **Bookmarks** - (Workspace Only) - Allows creation and management of [bookmarks](#).
- **Find** - (Workspace Only) - Opens window whereby specific [pipes and junctions can be found](#).

Related Topics

Workspace Window.....	384
Model Data Window.....	408
Output Window.....	413
Graph Results Window.....	426
Menus and Toolbars.....	359

Analysis Menu

- **Analysis Setup** - Opens the [Analysis Setup window](#).
- **Time Simulation ([XTS Module Only](#))** - Allows selection of either steady-state run or transient run.
- **Goal Seek & Control ([GSC Module Only](#))** - Allows selection of whether to use the [Goal Seek & Control](#) parameters in the current run.

- **Slurry (SSL Module Only)** - Allows selection of whether to use normal fluid calculation or settling slurry calculation in the current run.
- **NFPA Configuration** - NFPA must first be enabled in the [NFPA panel](#). Its use can then be toggled on and off here.
- **Automated Sizing (ANS Module Only)** - Allows selection of whether to use Network Sizing calculations in the current run.
- **Fluid Properties...** - Opens the [Fluid panel](#) in [Analysis Setup](#).
- **Transient Control... (XTS Module Only)** - Opens the [Simulation Duration panel](#) in [Analysis Setup](#).
- **Run Model...** - When the model is complete (as determined by the [Analysis Setup window](#)), the Run Model selection sends the model to the Solver for solution.

Related Topics

Workspace Window.....	384
Model Data Window.....	408
Output Window.....	413
Graph Results Window.....	426
Menus and Toolbars.....	359

Tools Menu

- **Scenario Manager** - Opens the [Scenario Manager](#) window.
- **Last Scenario** - Switches back to the previously loaded [scenario](#).
- **Generate Disconnected Scenario** - (Output Only) - Generates a scenario which is no longer connected into the [Scenario Manager](#) hierarchy and thus does not inherit changes from parents.
- **Compare Scenarios** - Quickly sets up a [comparison](#) between the current scenario and a related scenario.
- **Scenario Comparison Tool** - Opens the [Scenario Comparison Tool](#) window, allowing users to compare differences between any two or more scenarios in a model.
- **Output Control** - Opens the [Output Control](#) window.
- **Output Print Content** - (Output Only) - Configures what content can be printed from the [Output](#) window.
- **Output Language** - Selects the display language for certain phrases in the Output window, Graph Results window, and the labels in Workspace Layers.
- **Model Data Control** - (Model Data Only) - Opens the [Model Data Control](#) window.
- **Model Data Print Content** - (Model Data Only) - Configures what content can be printed from the [Model Data](#) window.
- **Select Graph Parameters** - (Graph Results Only) - Opens window to allow user to select which parameters to graph.
- **Quick Graph Formatting** - (Graph Results Only) - Opens window to allow user to change quick graph title and axis names, fonts, and scale values.
- **Design Alerts** - Opens the [Design Alerts](#) window.

- **Cost Application Manager** - Opens the [Cost Application Manager](#) window. Cost data can be applied to all pipes and junctions.
- **Cost Summary** - Opens the [Cost Summary](#) window.
- **Fluid Property Viewers** - Gives options to open either the [SteamCalc Viewer](#) or [Chempak Viewer](#). These are separate executables bundled with the software. Chempak Viewer requires a license.
- **Activate Modules** - Opens the [Modules panel](#) in Analysis Setup where optional add-on modules can be activated or deactivated.
- **eLicense Information** - Visible if AFT Fathom was opened using an eLicense - shows general information about the eLicense.
- **Keep Network eLicense Checked Out** - If a network eLicense is used, it can be kept Checked Out to this machine. This will prevent it from being lost when going offline, and prevent it from being accessible by other users. The license will be good for the Checkout Period - generally one week but depends on the specific company license.
- **Model Diagnostics** - Opens the [Model Diagnostics](#) window which provides troubleshooting information.
- **Custom Units** - Allows custom unit names to be enabled or disabled. Custom unit names can be defined in the User Options under the [Unit System](#) section.
- **User Options** - Opens the [User Options](#) window.

Related Topics

Workspace Window.....	384
Model Data Window.....	408
Output Window.....	413
Graph Results Window.....	426
Menus and Toolbars.....	359

Library Menu

- **Add Junction to Library** - (Workspace Only) - Prompts you to enter the name to be used in the Junction Library for the currently selected junction. After naming, [adds junction to the library](#).
- **Edit Junctions** - Opens [Edit Junctions window](#) where you can delete or rename entries in the Junction Library.
- **Edit Fluids** - Opens [Edit Fluids window](#) where you can add, delete, modify or review custom fluids.
- **Edit Pipe Materials** - Opens [Edit Pipe Materials window](#) where you can add, delete, modify or review custom pipe materials.
- **Edit Insulations** - Opens [Edit Insulations window](#) where you can add, delete, modify or review custom insulations.
- **Edit Fittings & Losses** - Opens [Edit Fittings & Losses window](#) where you can add, delete, modify or review custom pipe fittings and losses.

- **Edit Solids (SSL Module Only)** - Opens the [Edit Solids window](#) where you can add, delete, modify or review solids properties for slurry modeling.
- **Cost Library** - Opens the [Edit Cost Library](#) window where you can add, edit, or review cost libraries.
- **Energy Cost Library** - Opens the [Energy Cost Library](#) window to input time varying energy costs.
- **Suggest Pump** - Opens the Suggested Pumps window which allows the user to search libraries created in Fathom or the Intelliquip™ online database of pump manufacturers to find and select a pump (and the associated performance curve) based on the operating point. See the [Pump Sizing](#) topic for more information.
- **Library Manager** - Opens the [Library Manager](#) window.

Related Topics

Workspace Window.....	384
Model Data Window.....	408
Output Window.....	413
Graph Results Window.....	426
Menus and Toolbars.....	359

Arrange Menu

- **Primary Windows** - Allows you to configure the Primary windows in two panes in a horizontal or vertical orientation. The primary windows can be further rearranged by dragging the tabs into the center of the screen and adjusted by moving the tab into the desired position in the docking guide (see Figure 1).

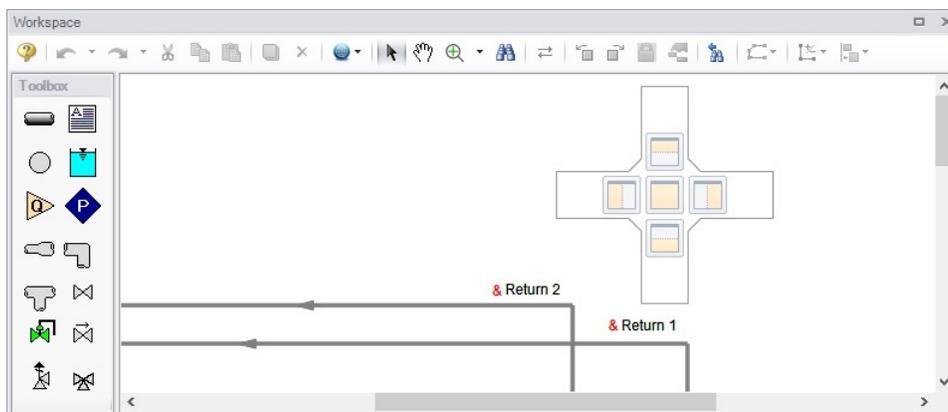


Figure 1: The Docking Guide can be used to view multiple primary window tabs at the same time

- **Sort Table By** - (Output Only) - Allows user to order tables in order of any variable and in ascending or descending order.
- **Rotate Icon Right or Left** - (Workspace Only) - Rotates icons for all selected junctions on the Workspace.

- **Auto-rotate Icon** - (Workspace Only) - Rotates the selected icon(s) to align with the connected pipes if possible.
- **Rotate Selection Right or Left** - (Workspace Only) - Rotates selected pipes and junctions as a group.
- **Align** - (Workspace Only) - Aligns selected pipes and/or junctions on the Workspace. You can choose to align pipes and junctions, just pipes, or just junctions horizontally or vertically on the workspace.
- **Pipe Drawing Mode** - (Workspace Only) - Select the type of grid layout that the model will use.
 - **2D Freeform** - Junctions can be placed anywhere on the Workspace.
 - **2D Orthogonal** - Toggles orthogonal drawing capability on the Workspace. This toggle is a shortcut to enabling the feature in [User Options](#) window.
 - **Isometric** - Uses an isometric grid.
- **Snap to Grid** - (Workspace Only) - Available for 2D grids (on by default for isometric grid). Toggles Snap to Grid feature on the Workspace. When ON, all Workspace pipes, junctions and annotations snap to a grid specified in [User Options](#) window.
- **Show Grid** - (Workspace Only) - Toggles Show Grid feature on the Workspace. Shows or hides grid that is specified in [User Options](#) window.
- **Make Selected Pipes and Jcts Orthogonal** - (Workspace Only) - Makes all selected pipes and pipes connected to selected junctions on the Workspace orthogonal.
- **Flip Vertex** - (Workspace Only) - Available when orthogonal drawing is enabled. This will flip the vertex of all selected pipes on the Workspace.
- **Scale/Flip Workspace** - (Workspace Only) - [Scales \(shrinks or expands\) or flips](#) either selected or all pipes and junctions on the Workspace.
- **Set Pipe and Junction Locations to Parent** - (Workspace Only) - Sets locations of pipe and junction labels to the same position as the parent [scenario](#).
- **Reset Label Locations** - (Workspace Only) - Sets locations of pipe and junction labels to their default locations.
- **Disable Pipes Snapping to Isometric Grid** - (Workspace Only) - Toggles Snap to Grid feature on the Workspace for the selected pipe. When Snap to Grid is disabled for that pipe, the pipe end point no longer snaps to grid points and pipe segments do not follow the [Isometric](#) lines.
- **Add/Remove Segments** - (Workspace Only) - Allows user to Add Segment (visual bend in pipe), Add Segment at Right Angle, Add Multiple Segments, Remove Segment, or Remove all Segments.
- **Lock Object** - (Workspace Only) - Locks selected pipes and junctions on the Workspace so they cannot be moved. A lock symbol appears next to locked pipes or junctions when an attempt is made to move the pipe or junction.
- **Reverse Direction** - (Workspace Only) - Reverses the [reference positive flow direction](#) of all selected pipes on the Workspace.

Related Topics

Workspace Window.....	384
Model Data Window.....	408

Output Window.....	413
Graph Results Window.....	426
Menus and Toolbars.....	359

Window Menu

- **Workspace** - Makes the [Workspace window](#) active.
- **Sizing (ANS Module Only)** - Makes the [Sizing window](#) active.
- **Model Data** - Makes the [Model Data window](#) active.
- **Output** - Makes the [Output window](#) active.
- **Graph Results** - Makes the [Graph Results window](#) active.

Related Topics

Workspace Window.....	384
Model Data Window.....	408
Output Window.....	413
Graph Results Window.....	426
Menus and Toolbars.....	359

Help Menu

- **Help File** - Opens help site home page
- **Use Offline Help** - Indicates that the PDF help file will be used instead of the Online AFT Help File when the help buttons in topics are clicked. Note that the PDF help file must be downloaded from the PDF Downloads page on the online help site prior to using this option.
- **Quick Start Guide** - Opens quick start guide pdf in English or metric units
- **Quick Start Video** - Opens quick start video series on aft.com
- **Optimization Assistant (ANS Module Only)** - Opens the Optimization Assistant to help identify input that will potentially effect the sizing results. Also appears if problems are identified when the model is run.
- **Keyboard Shortcuts** - Opens [keyboard shortcut](#) help page in help system
- **Show Examples** - Opens help system with links to various example problems. Note that the Examples PDF must be downloaded from the PDF Downloads page on the online help site if the Use Offline Help option is active.
- **Video Tutorials** - Opens video tutorials on aft.com in English or metric units and English or Spanish language
- **Technical Seminars** - Opens information on training courses on aft.com.
- **Flow Expert Package** - Opens information on aft.com for individualized help from AFT engineers for training, modeling, consulting, and more
- **Online Learning Center** - Opens learning center on AFT website with links to resources such as blogs, video tutorials, webinars, etc.
- **AFT Home Page** - Opens home page on aft.com

- **AFT Blog** - Opens blog home page on aft.com
- **Send Email to Support** - Links to AFT support in your default email program
- **Latest Release** - Opens page on aft.com to download the newest AFT Fathom release
- **AFT Notifications** - Opens page on aft.com where you can sign up for email notifications for new software updates and releases
- **About AFT Fathom** - Opens window in AFT Fathom with information on current AFT Fathom version, license number, etc.

Related Topics

Workspace Window.....	384
Model Data Window.....	408
Output Window.....	413
Graph Results Window.....	426
Menus and Toolbars.....	359

Toolbars

The Toolbars offer quick access to the features used most frequently in AFT Fathom. There are actually several Toolbars in all. One is the Common Toolbar, and it is always shown. Its features are common to all Primary Windows. In addition, there is one Toolbar for each [Primary Window](#), offering features appropriate for work in that window. You only see two Toolbars at a time in AFT Fathom. The function of each icon in a given Toolbar appears when you pass the mouse over the icon button without clicking on the button. Each of the functions on the Toolbar is also available through the menu system.



Figure 1: Common Toolbar



Figure 2: Workspace Toolbar



Figure 3: Model Data Toolbar



Figure 4: Output Toolbar



Figure 5: Graph Results Toolbar

Related Topics

Workspace Window.....	384
Model Data Window.....	408
Output Window.....	413
Graph Results Window.....	426
Menus and Toolbars.....	359

Printing

Printer Page Layout

The Printer Page Layout tool enables users to control how a model on the Workspace will be split across pages in the Print Preview window. To access this tool click the printer icon in the bottom-right of the Workspace. The Printer Page Layout options should appear.

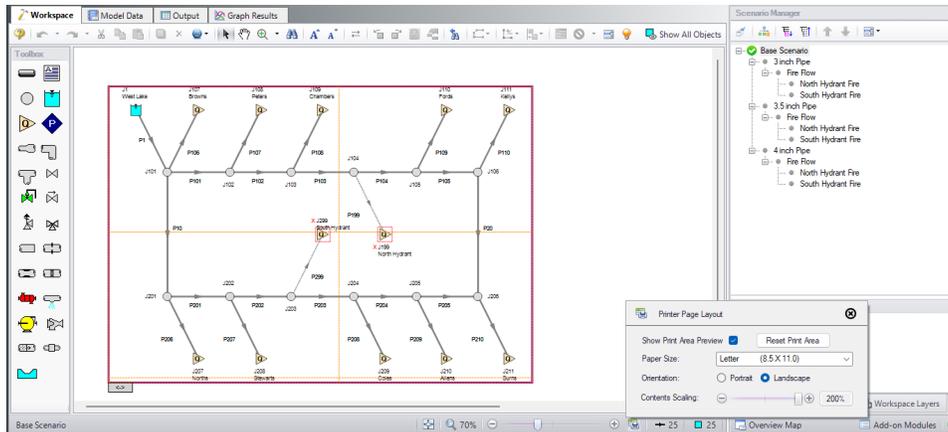


Figure 1: The Printer Page Layout options control how the model is split between pages on the Print Preview

- Show Print Area Preview** - When checked, a purple box will appear on the Workspace which dictates what areas will be printed. Clicking on this purple box will cause sizing handles to appear which can be used to re-size the area. Additionally, the gray box in the lower-left of the purple print area can be used to drag and re-position the area relative to the objects on the Workspace. The purple print area is affected by the Snap to Grid option from the Arrange menu. Furthermore, dotted yellow lines may also be drawn within the purple print area depending on the other settings described below. The dotted yellow lines represent different pages that will be printed. If the

dotted yellow lines form a single box around the model, this means the entire model will fit on a single page. If the dotted yellow lines form multiple boxes across the model, regardless of the size of the boxes, the model will be divided such that everything in a single yellow box will appear on its own page once the Print Preview is used.

- **Paper Size** - The paper size is a drop down with a selection of page dimensions based on the page sizes. The list of page sizes is determined by whatever printer page sizes are supported by your printer. Selecting one of these sizes will affect the size of the paper the model will be printed on. As such, the size of the boxes formed by the dotted yellow lines from the "Show Print Area Preview" will adjust in size.
- **Orientation** - The portrait setting will flip the page such that the longest side from the paper size will align vertically on the page. The Landscape setting will flip the page such that the longest side from the paper size will align horizontally. As such, the boxes created from the "Show Print Area Preview" option will follow suit.
- **Contents Scaling** - This feature determines how large the model will appear on each page. As the slider moves, it will appear as though the dotted yellow boxes from the "Show Print Area Preview" are shrinking or growing; however, this does not affect the dimensions of the pages. Instead, when the Print Preview is used the page size will remain the same, but the objects in each box will fit to the pages as displayed on the Workspace.

Print Preview

The Print Preview window allows you to access special print features such as Print Preview and Fit to One Page. It is accessed from the [File menu](#) or using the shortcut CTRL + P.

The Print Preview window also offers you a preview of the print content and format for all [Primary Windows](#). By clicking the Print Preview button you can review your printed material at several zoom states. Individual pages can be printed.

The "Fit To Page(s)" feature causes the graphic image generated from the Workspace window to be compressed onto the specified number of pages. This is a convenient feature for creating reports of your model. The Print Preview supports whatever printer page sizes are supported by your printer, so the use of legal paper or larger may allow you to get more visible details. The graphics can also be centered vertically and/or horizontally. You also can fit your content onto multiple pages.

The Page Setup option in the Print Preview window allows you to set the page orientation, paper size/type, and margins.

Additional special printing features affect the Workspace. The "Selected on Workspace" option works with Workspace printing by selectively printing the pipes and junctions currently selected on the Workspace.

By default, the junction icons on the Workspace will be printed with the same colors as displayed on the screen. It also prints the background color as specified in the User Options > Workspace > Colors and Fonts. You can also specify that the junction icons be printed in black and white (still with the color background) or that the entire printout be black and white.

If you added a background picture from the [File menu](#), this picture will be included in the printout.

Finally, if you have a grid displayed on the Workspace, the grid will be included in the printout.

Page options allow the use of a company logo, user comments, titles, and different options for borders.

Related Topics

Interface and User Options.....	359
---------------------------------	-----

Related Examples

Beginner - Three Reservoir	823
----------------------------------	-----

Primary Window Overview

AFT Fathom has several Primary Windows that work in an integrated fashion. Each is located on a separate, moveable tab. You work exclusively from one of these windows at all times. For this reason, they are referred to as *Primary Windows*.

You can change between the Primary Windows by selecting them from the [Window menu](#) or by clicking the appropriate tab within the AFT Fathom window.

- [Workspace](#)

The Workspace Primary Window is where the user creates the graphical representation of the system. Engineering data is input on pipe and junction property windows which are opened by double-clicking the objects.

The Workspace Layers are also located on this Primary Window and allow the user to display both input and output data superimposed on the pipe system schematic.

- [Model Data](#)

The Model Data Primary Window summarizes the engineering input data from the Workspace into categorized tables. This allows for easy review of the system inputs, and double clicking cells will open the pipe or junction properties for quick editing.

- [Output](#)

The Output Primary Window offers a powerful and diverse range of features for reviewing analysis results which are organized into categorized tables and lists. The Output allows the user to easily compare multiple design scenarios with color coded rows, details errors or design requirement violations, and prepare the results for documentation.

- [Graph Results](#)

The Graph Results Primary Window allows the user to create a wide array of graphs with various parameters which are integral to gaining a deeper understanding of the pipe system's behavior. The user can adjust various formatting settings such as axis ranges, labels, colors, and create dual y-axis or stacked graphs. The settings can be saved into templates called Graph List Items so that the graphs can be easily reproduced.

If the [ANS Module](#) is activated, an additional Primary Window called the [Sizing window](#) will be available, which allows sizing information to be configured.

Primary Window	Can Display Input	Can Display Output
Workspace	✓	✓
Model Data	✓	
Output		✓
Graph Results		✓

Related Topics

Interface and User Options..... 359

Related Examples

Beginner - Three Reservoir 823

Status Bar

AFT Fathom's Status Bar displays contextual information about the current scenario or operation.

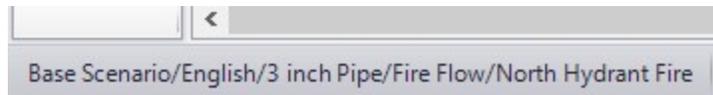


Figure 1: Current scenario displayed in the Status Bar

The Status Bar indicates the status of certain operations - for example, when opening a model the progress of loading the pipes, junctions, and scenarios is displayed.

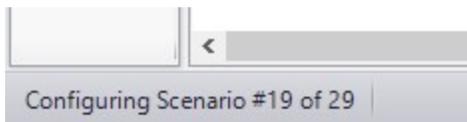


Figure 2: Status Bar displaying progress when opening a model

Related Topics

Related Examples

Quick Access Panel

The Quick Access Panel, as shown in Figure 1, contains several useful features:

- **Scenario Manager** - Configure the scenarios and variations of the model
- **Properties tab** - Review pipe/junction inputs or output, and edit annotations
- **Workspace Layers** - Superimpose input and output parameters on top of the Workspace model
- **Overview Map** - Easily navigate large models
- **Activate Add-on Modules** - Shortcut to the Add-on Modules group in the Analysis Setup
- **Model Status Light** - Green/red light indicating if all required input is present and if the Solver is ready to run

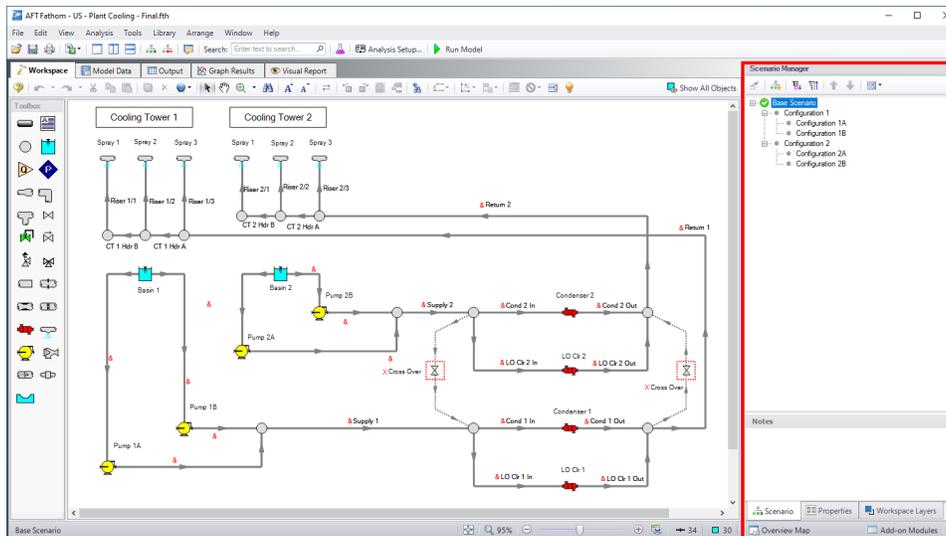


Figure 1: The Quick Access Panel

Scenario Manager

By clicking on the Scenario tab in Figure 2 below, you have the full scenario customization ability as you would with the [Scenario Manager](#). You can create child scenarios, expand or collapse various families of scenarios, and use the many other Scenario Manager features here as well.

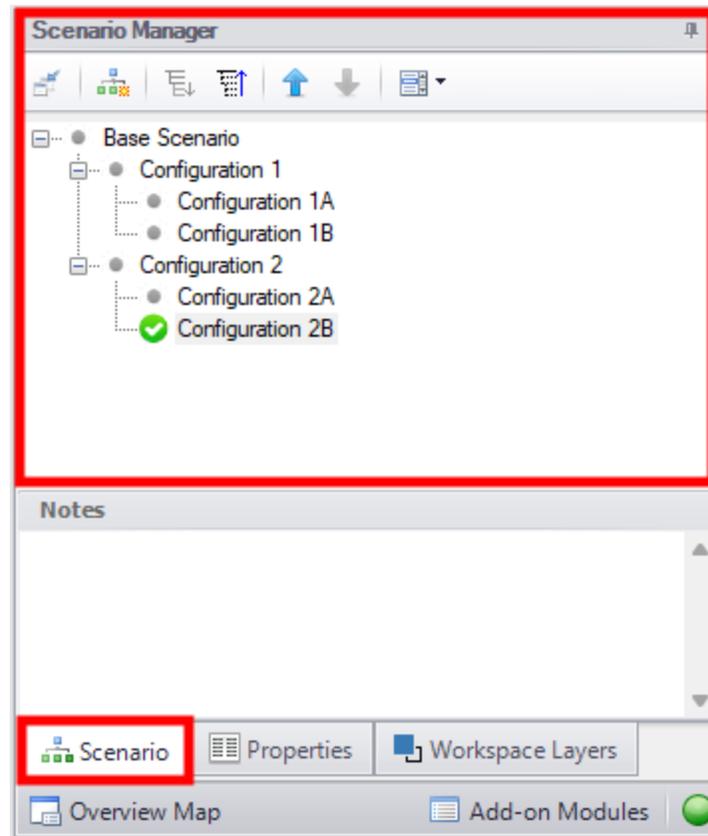


Figure 2: The Scenario Manager

Properties Tab

As seen in Figure 1 or 2, there is a "Properties" tab in the Quick Access Panel. When pipes and junctions are drawn on the Workspace, you can quickly view the Pipe or Junction Properties by first selecting a pipe or junction on the Workspace, then clicking on the "Properties" tab. The tab's name will dynamically update depending on which pipe/junction is selected. This will display the Object Property data as well as any Output data if the model has been run. Figure 3 below shows an example of what the Properties tab would look like when clicking on a pipe on the Workspace after it has been defined and output was created.

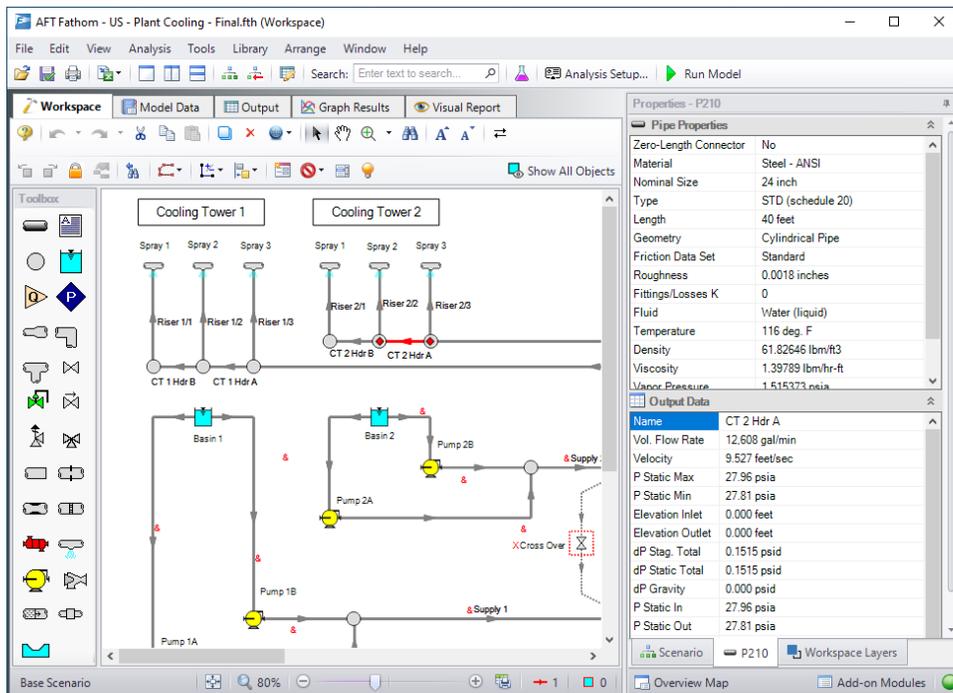


Figure 3: Viewing the Pipe/Junction Properties and Output Data from the Quick Access Panel

[Annotations](#) can be edited by selecting an annotation on the Workspace which will bring up their properties in this tab where they can then be edited.

Interface and User Options

Properties - Annotation 1

Enter text to search... Defaults ?

Annotation (ID = 1)

Name	Annotation
Text	Group B ...
Text Alignment	TopLeft
Text Color	Black
Text Font	Microsoft Sans Serif, 18pt ...
Text Show	<input checked="" type="checkbox"/>

Behavior

Anchor Object Tag	0
Anchor Type	Workspace
Workspace Allow Select	<input checked="" type="checkbox"/>
Workspace Keep In Back	<input type="checkbox"/>

Format

Border Color	Black
Border Style	Solid
Border Thickness	1
Shape	Rectangle 1:1
Shape Color	White
Shape Show	<input checked="" type="checkbox"/>

Image

Image	...
Image Keep Aspect Ratio	<input type="checkbox"/>
Image Stretch	<input checked="" type="checkbox"/>

Line

Arrow At End	<input checked="" type="checkbox"/>
Arrow At Start	<input type="checkbox"/>
Arrow Size	3
Arrow Style	Fill
Line Color	Black
Line Show	<input type="checkbox"/>
Line Start Location	TopLeft
Line Style	Solid
Line Thickness	1

Scenario Annotation 1 Workspace Layers

Figure 4: Annotation view for the Properties in the Quick Access Panel

Workspace Layers Tab

The [Workspace Layers](#) tab contains settings relating to object visibility, labels, superimposing input/output data, and color maps.

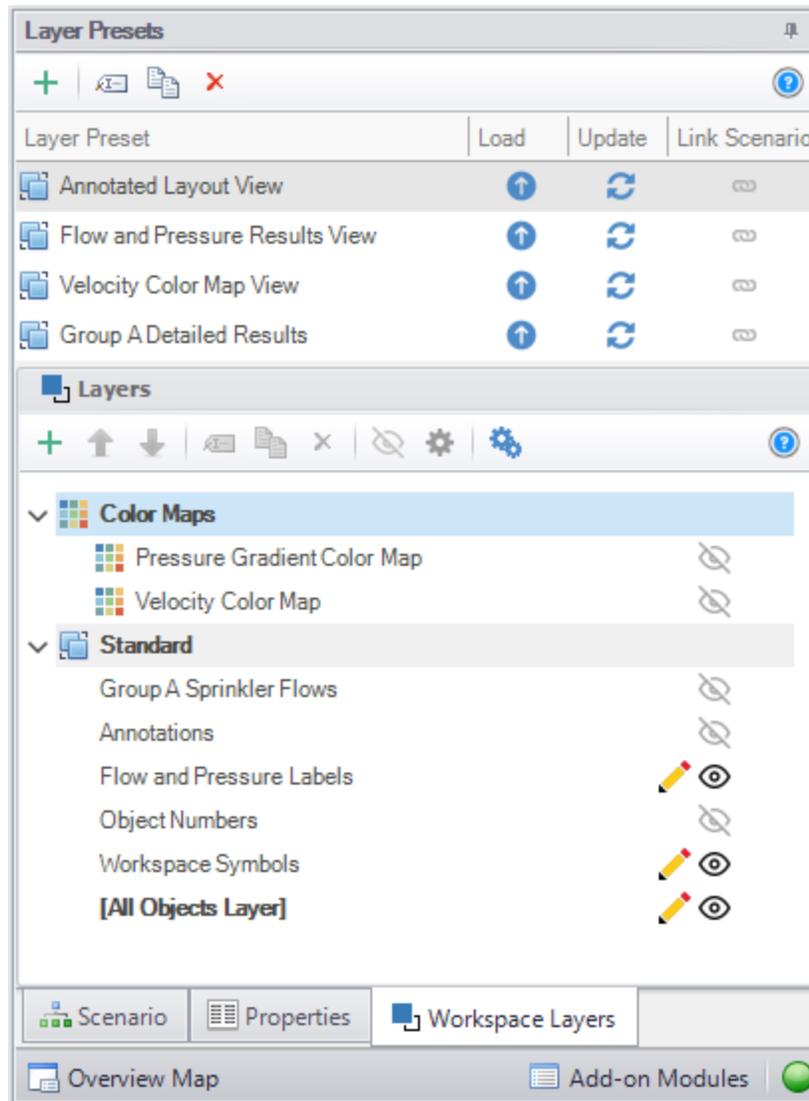


Figure 5: View of the Workspace Layers panel structure

Overview Map

When a model is very large in size, it can be troublesome to have to constantly drag the model around on the Workspace to get to a specific area of the model you want to view. By clicking on the Workspace Overview Map button in the red box in Figure 6, you will bring up a map that is a bird's eye view of the model. Drag the small square in the Workspace Overview Map over various parts of the model, then zoom in and out of the model. You can also click the "Fit" button to see the full model zoomed out on the Workspace. You can also zoom in and out using the mouse wheel.

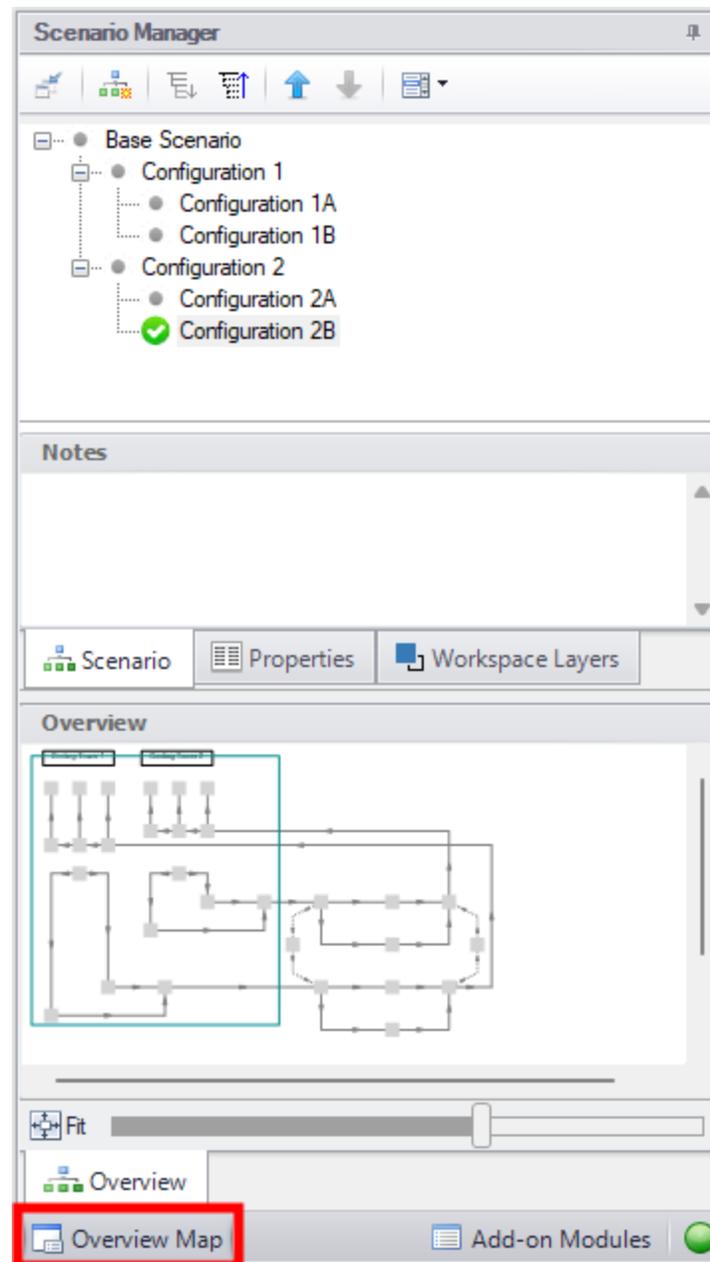


Figure 6: Workspace Overview Map

Activate Add-on Modules

Clicking the Modules button in the bottom right of the Quick Access Panel in Figure 7 launches Analysis Setup to the Modules panel to activate or deactivate add-on modules.

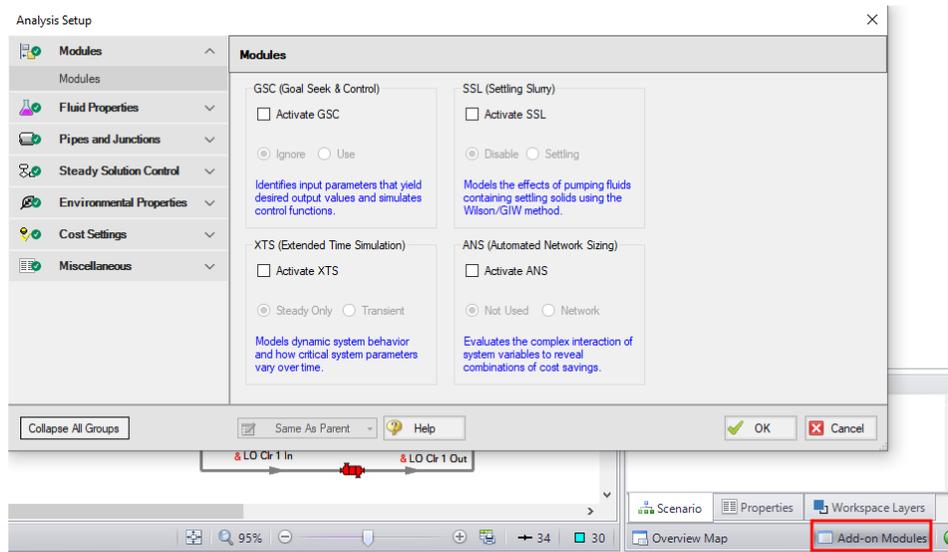


Figure 7: Activating the Modules from the Quick Access Panel

Model Status Light

The Model Status Light is located in the bottom-right of the Quick Access Panel (Figure 8). Clicking the Status Light will open Analysis Setup. When each item in Analysis Setup is fully defined, it will have a green check mark. When all necessary items in Analysis Setup are completed, the Status Light will turn from red to green, indicating that the model can be run.

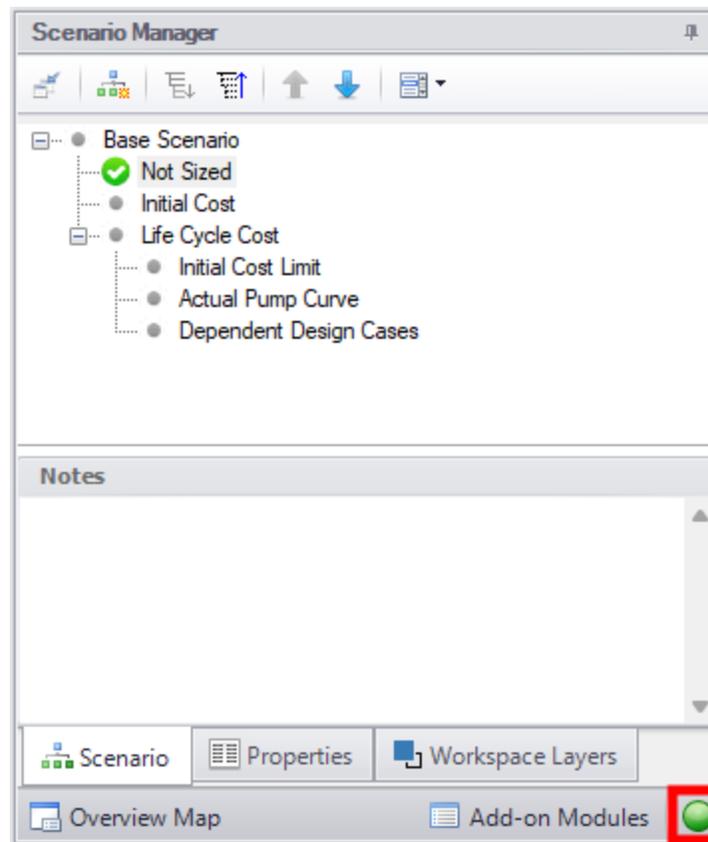


Figure 8: The Model Status Light is located at the bottom-right of the Quick Access Panel

Minimizing the Quick Access Panel

By default, AFT Fathom opens with the Quick Access Panel automatically expanded. The Quick Access Panel can be easily collapsed and pinned to the side of the Workspace to allow for a larger usage area of the Workspace. Figure 9 shows the AFT Fathom window with the Quick Access Panel initially expanded. By clicking on the thumbtack pin button (highlighted below with the red box), the Quick Access Panel will then become collapsed as shown in Figure 10. After the Quick Access Panel is collapsed, it can be temporarily expanded again simply by hovering over the "Scenario" sidebar with the mouse.

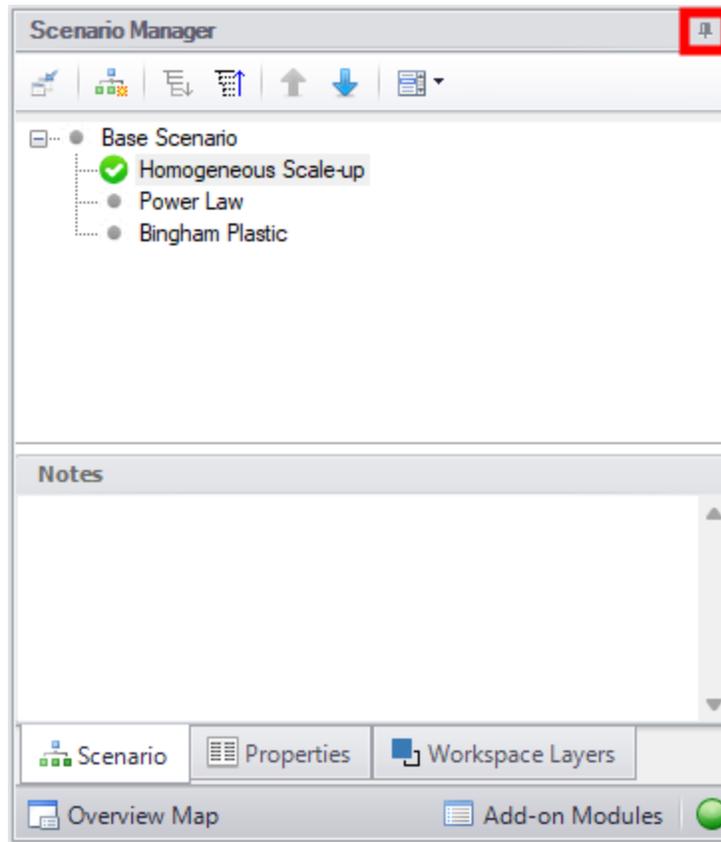


Figure 9: The Workspace with the Quick Access Panel expanded



Figure 10: The Workspace with the Quick Access Panel collapsed

Related Topics

Interface and User Options.....	359
---------------------------------	-----

Related Examples

Beginner - Three Reservoir	823
----------------------------------	-----

Model Status Light

The light at the right of the Status Bar located at the bottom of the AFT Fathom window, which indicates whether or not the model can be sent to the Solver (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system).

Clicking on the Model Status Light will show the [Analysis Setup window](#), which indicates incomplete inputs with a red exclamation mark: 

Related Topics

Interface and User Options.....	359
---------------------------------	-----

Related Examples

Beginner - Three Reservoir	823
----------------------------------	-----

Solution Progress Window

Solution Progress indicates the current state of the solution iterations.

The most out of [tolerance](#) junction or pipe is listed for the appropriate tolerances, along with general information about the solution method and related settings.

Note: Both absolute and relative criteria are always displayed. This can be helpful for selecting the appropriate tolerance criteria on difficult to converge models.

The area in the bottom of the window will give important feedback on the model solution. It is important not to dismiss messages in this area, as they could have an impact on results.

When the model solution is converged, options to view [Output](#), [Graph Results](#), and [Workspace](#) become available.

Pausing the iterations allows the modification of certain [Steady Solution Control](#) parameters during the run, as well as viewing the iteration history. These options are available under Other Actions.

Related Topics

Interface and User Options.....	359
Iterations Panel.....	165
Review the Iteration History.....	804

Related Examples

Beginner - Three Reservoir	823
----------------------------------	-----

Workspace Window

The Workspace window is the sole area where you specify the model layout and connectivity. The Workspace window has two separate functional areas: the [Toolbox](#) on the far left and the Workspace itself encompassing the rest of the screen (see Figure 1).

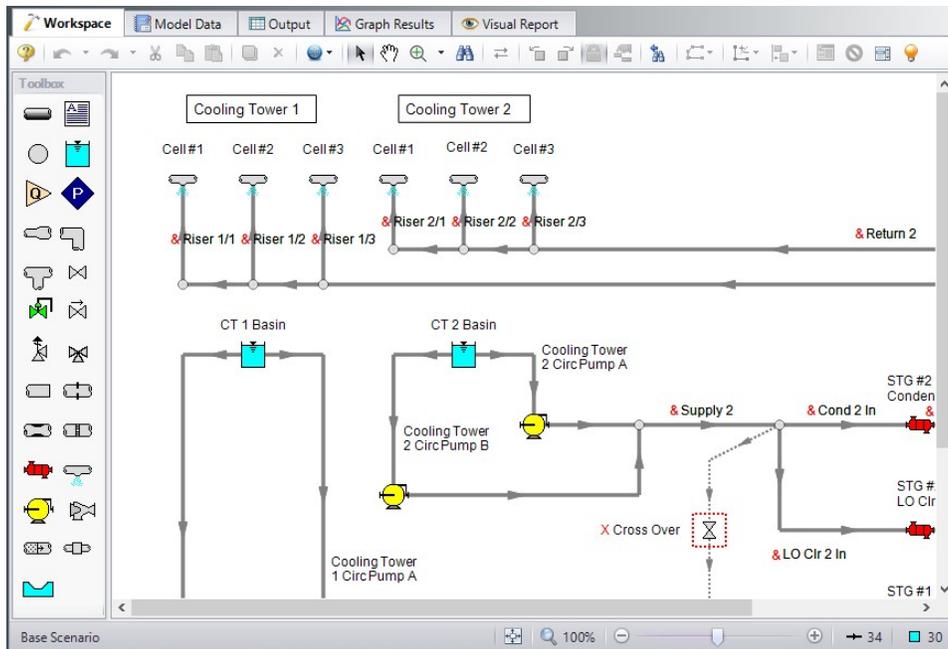


Figure 1: The Workspace window

Printing

The Workspace image can be:

- Copied to the clipboard (Note: cannot be done from print preview)
- Saved to a picture file
- Saved directly to an Adobe PDF file
- Sent to the printer

The [Print Preview](#) window allows the above outputs to be applied to all or certain selected Workspace objects.

Related Topics

Interface and User Options.....	359
Menus and Toolbars.....	359
Workspace Layers.....	385
Annotations.....	395
Merging Models.....	400
Renumbering Workspace Objects.....	399
Scale/Flip Workspace.....	401
Magnifier Tool.....	401
Extended Model Check.....	406
Keyboard Shortcuts for Workspace.....	404
Undefined Objects Window.....	403
Raw Data Changes.....	407
Duplicate Special.....	402

Related Examples

Beginner - Three Reservoir.....	823
---------------------------------	-----

Workspace Layers

Fundamentally, Workspace Layers are used to control the visibility of [Workspace](#) objects and labels. Layers allow custom views to be created in which results can be displayed without needing to leave the Workspace window. A group of layers functions like a stack of transparencies, with each layer displaying specific, customizable information. The view on the Workspace is a composite of the information presented by all visible layers, but all objects exist regardless of visibility for the purposes of calculations and object connections.

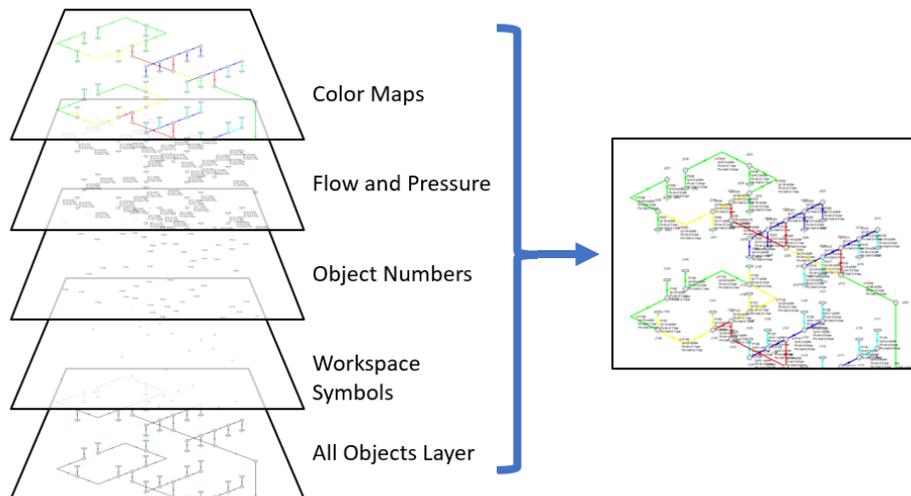


Figure 1: Information from all currently visible layers combines to form the view displayed on the Workspace

Layers can be shown or hidden to create different views of the Workspace. The visibility of objects on the Workspace depends on the layers that are shown, as objects on hidden layers are not shown on the Workspace unless present on another layer. If an object exists on multiple layers, only the top-most representation is visible. The order of the layers is controlled by the order in which they are listed in the Workspace Layers panel.

Workspace Layers Panel

The Workspace Layers panel is accessed via the Workspace Layers tab in the [Quick Access Panel](#). There are two areas within the Workspace Layers panel to control Layer Presets and Layers.

In the Layers area, new layers can be created, and any selected layers can be renamed, duplicated, and toggled in visibility. The Global Layers Settings menu is accessed here, and the column of visibility icons on the right shows the current visibility of each layer and can be clicked to toggle visibility. Hovering over a layer, a gear icon appears that links to the layer's Layer Settings options for customization.

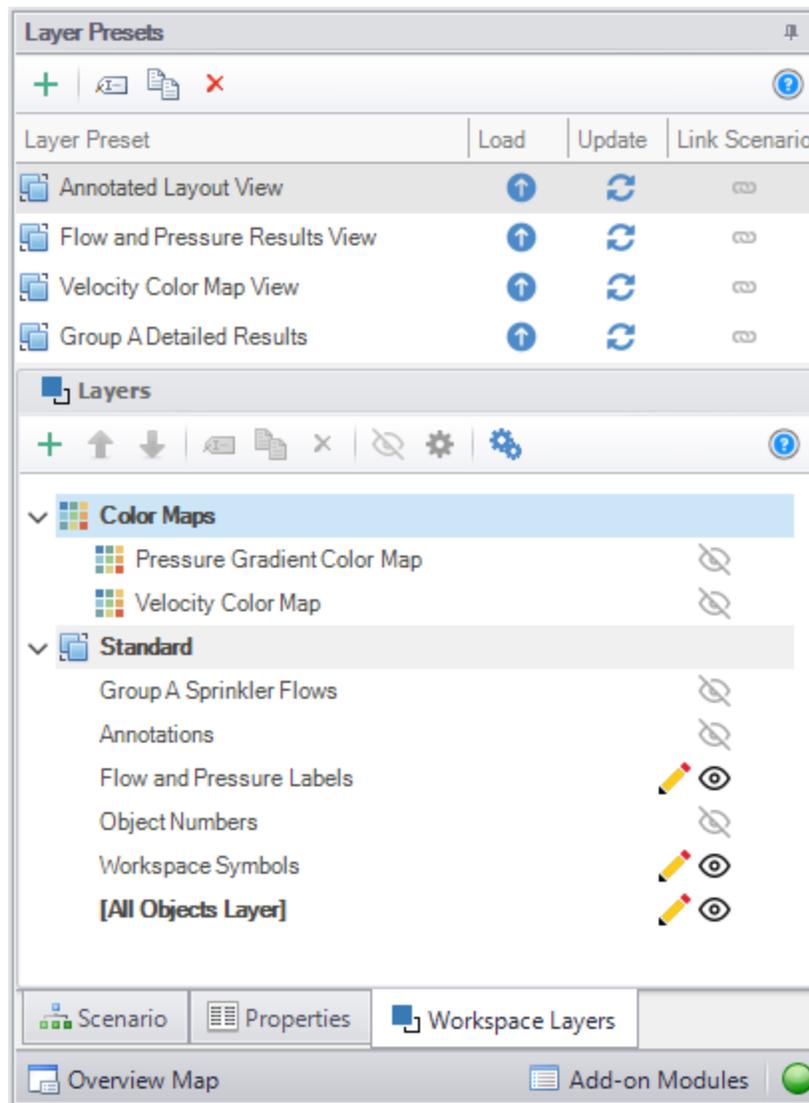


Figure 2: View of the Workspace Layers panel structure

Standard layers control pipe and junction styles, labels, and visibility, along with the visibility of any annotations in the model. Among the standard layers is the All Objects Layer. The All Objects Layer exists in all models and contains all pipes, junctions, and annotations in the model. This serves as the primary layer, provides a central location for users to quickly see all objects in the model, and is always the bottommost layer. When creating a view where some objects are hidden, the All Objects Layer can be toggled in visibility so that only objects visible in other layers are shown on the Workspace.

Other standard layers can be reordered up or down the layer stack using the areas at the top of the Layers area. For pipe and junction labels, the order of displayed text items is affected by the order of the layers containing label text.

Standard layers can be created from the New Layer drop-down as a Blank Layer or From Workspace Selections. Layers created from Workspace selections have the selected objects and their labels visible in the new layer.

By default, new objects are visible only on the All Objects Layer. However, if layers are set as Active Layers, then any new objects that are created and their labels will be visible within all Active Layers. Active Layers are set by hovering over a layer and clicking the pencil icon and any currently Active Layers are denoted by the pencil icon.

Layer Presets

Layer Presets allow customized Workspace views to be retained by saving the status (visibility) of each layer. There are four columns in the Layer Preset area. One shows the name of the preset, the Load Preset option allows the previously saved layer states to be loaded onto the Workspace, the Update Preset option saves the current layer states to that Layer Preset, and the Link Preset option links that preset to the currently active scenario.

By linking a preset to a scenario, it will automatically load whenever that scenario is loaded in the future. Individual layers, by contrast, are not tied to a specific scenario and are defined globally and immediately available in all scenarios. A layer can only show objects that exist in the active scenario, and changes to the style of an object in a layer will affect all scenarios.

Global Layer Settings

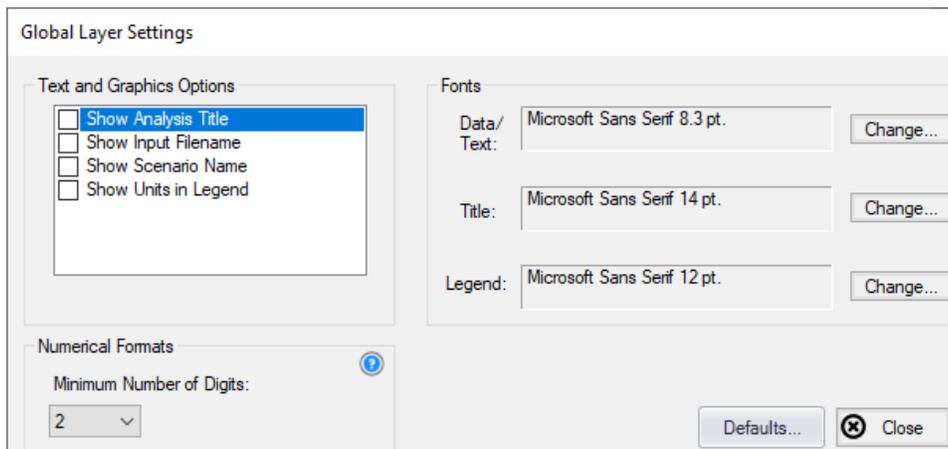


Figure 3: Global Layer Settings window

The Global Layer Settings are accessed from the double gear icon on the Layers Toolbar in the Quick Access Panel. The properties that are controlled by Global Layer Settings are not defined for individual layers – changes to these settings affect all layers globally.

Text and Graphics Options allow the analysis title, input filename, scenario name, and units in the legend to be toggled.

Note: The analysis title is set in the general tab of [Output Control](#).

(XTS Module Only) For using [XTS in the Workspace](#), the Show Time for Displayed Results option shows the time associated with the current transient results on the Workspace. This is particularly useful for printing.

The Fonts setting controls the fonts for Data/Text (e.g. labels), the Title text (the analysis title, input file-name, and scenario title if visible) and the text in the legend.

The Numerical Formats option determines the minimum number of digits shown for input and output data displayed within label text.

Layer Settings - Pipes & Junctions

The Layer Settings window can be accessed by double-clicking the layer, from the single gear icon on the right of each layer, or by selecting the layer and clicking the single gear icon on the Layers Toolbar.

The layout of the Layer Settings window is similar to Analysis Setup, with a number of *groups* listed on the left side of the window. Each group contains *items* that when selected change the content of the main panel area on the right.

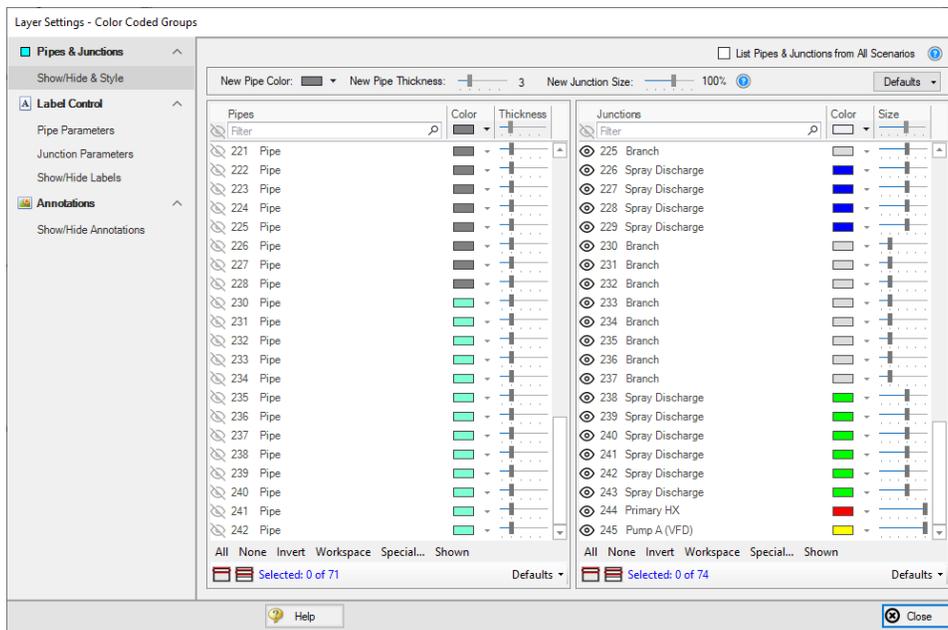


Figure 4: The Show/Hide & Style panel allows the appearance and visibility of pipes and junctions to be managed

The Pipes & Junctions group of the Layer Settings window allows the pipe and junction color and size/thickness to be defined for a given layer, along with which pipes and junctions are visible. Note that changes to this panel are made live in real time, and no Apply button is needed to apply the changes that you make. Pipes and junctions are shown in two separate lists, which can be filtered.

Multiple pipes or junctions can be edited at a time using the Color, Size, and Thickness modifiers at the top of each table, or set to default options with the Defaults buttons towards the bottom. The usual tools for selecting pipes and junctions, including Workspace select and [Select Special](#), are available here. Pipes and junctions may also be edited individually with the visibility, color, and size options in each row.

The appearance of new pipes and junctions is layer specific, and can be set to default options with the top Defaults button or modified with the New Pipe Color, New Pipe Thickness, and New Junction Size options. Note that color and icon style for new junctions are set via the [Toolbox](#) as they are specific to each type of junction.

List Pipes & Junctions from All Scenarios is toggled off by default to reduce unintended changes to items in other scenarios, but this can be toggled on to include objects from all scenarios in the lists.

Layer Settings - Label Control

A label is text that is associated with a pipe or junction on the Workspace. Labels include both input and output parameters along with object names, numbers, and workspace symbols. While there is no separate output mode, output parameters are only displayed when the current scenario has output data available. If no output is present, the parameter in the list to the right will appear grayed out. Input parameters will always be shown when added to the list on the right. The label shown for an object on the Workspace is a combination of defined label items from each visible layer, and the order of the items is determined by both the layer order and the order that parameters are listed within each layer.

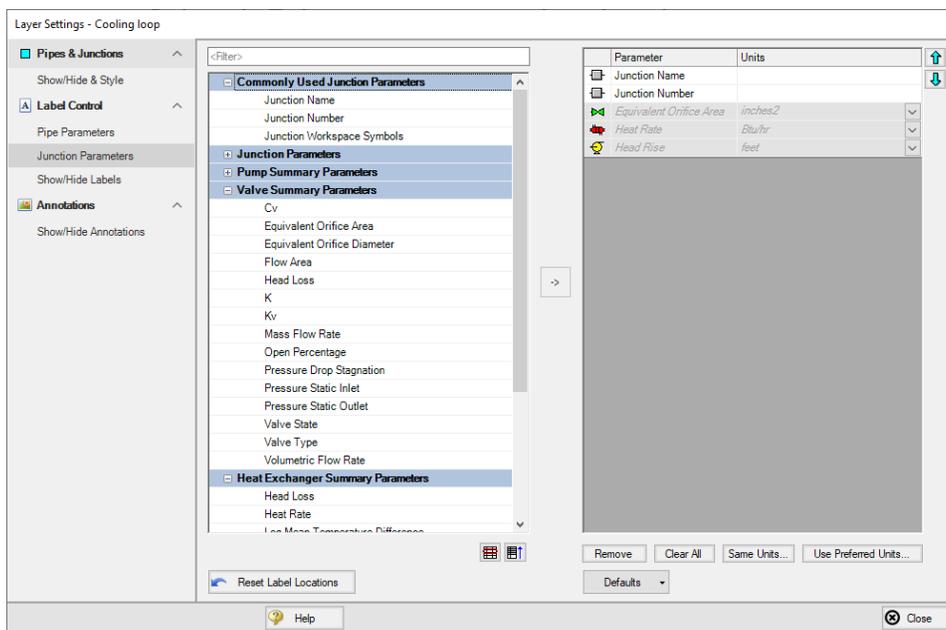


Figure 5: The Junction Parameters panel within the Label Control group of Layer Settings

Within the Label Control group, the Pipe Parameters and Junction Parameters items allow for the parameters associated with a layer to be customized. When there is no output available, output parameters will be grayed out within the list, but their list position and units may still be edited.

Label locations are defined globally, so the Reset Label Locations option will apply to all labels in all layers and all scenarios.

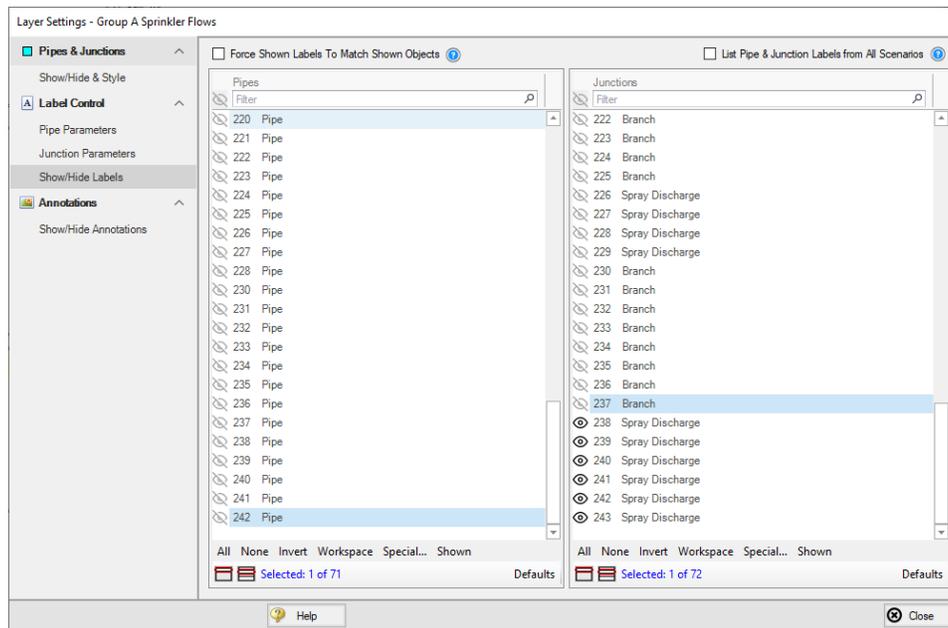


Figure 6: The Show/Hide Labels panel determines which objects will have visible labels for a given layer

The Show/Hide Labels panel can be used to toggle the visibility of labels for each pipe and junction within a layer. This allows a user to create a layer that shows only the relevant parameters for the relevant objects in the model. Visibility can be modified for one or several objects at a time via the same methods as pipe and junction visibility.

The Force Shown Labels to Match Shown Objects option forces the visibility of each label to match its object visibility as defined in the Pipes & Junctions group of the Layer Settings. The Defaults options at the bottom of the tables simply toggle this option on if it had been previously unselected.

Layer Settings - Annotations

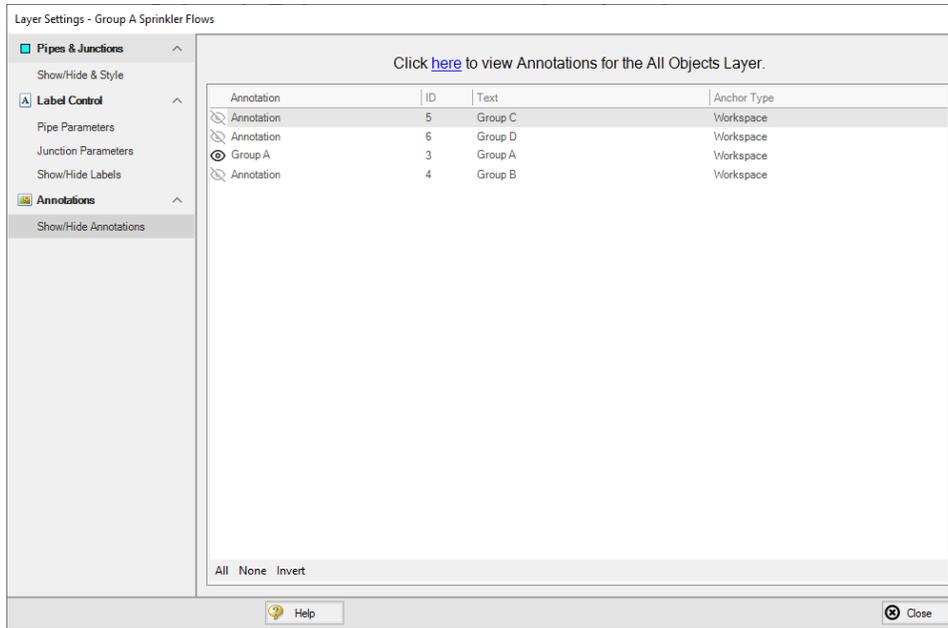


Figure 7: The Show/Hide Annotations panel shows all annotations that exist in the model file and allows their visibility to be controlled on a per-layer basis

The Show/Hide Annotations panel allows the visibility of annotations to be defined for the current layer and displays the visibility of each annotation in the current layer. While annotations exist globally, in concert with Layer Presets this menu allows annotations to be controlled so that they are only visible in the relevant scenarios. Annotations can be edited from this list view by double-clicking them in the list to open the annotation in the Annotation Editor.

Color Maps

A Color Map layer allows you to visualize magnitudes of various parameters by coloring the pipes. Color Maps are a special layer type, and are shown in a separate group on the Layers panel. There can be only one special layer visible at a time, and they override the behavior of standard layers in the model - in this case superseding any colors assigned to pipes by other layers.

Example use cases for Color Maps includes visualizing hot/cold loops, locations of high velocity/reverse flows, or pipe diameters/lengths.

To create a Color Map layer, click the new layer icon on the Layers Toolbar, select a Color Map layer type, and name the new layer. To edit the layer, go to its Layer Settings menu, accessed in the same manner as a standard layer.

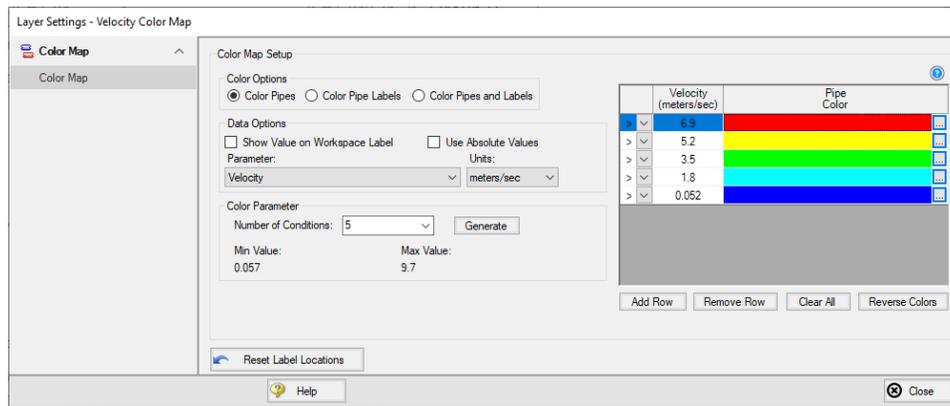


Figure 8: Layer Settings window for a Color Map

In the Color Map Layer Settings window, choose a parameter and units for which all pipes will be categorized. Then choose the Number of Conditions (number of colors) and click Generate. The operators (i.e. >, >=, =, <=, <) and values can be modified. Values may be categorized on an absolute value basis by selecting the Use Absolute Value checkbox. The Show Value on Workspace Label option shows the evaluated parameter next to each visible pipe in the model, and combines with any other label text items from other layers.

The order of entry is not important. Fathom will sort the entries first by value and then by the operator, with the greater than (>) operator being above the less than (<) operator. The pipes will be colored using the first match found in the map, starting from the top of the map. If no match is found, the pipe will be the Workspace pipe color. Be careful to avoid overlapping operators and values. If, for example, the map defined four colors using the following inequalities,

- >=60
- <60
- >=20
- <20

then there is no way colors 3 and 4 will be used because any pipe less than 20 is also less than 60. Since <60 is before <20 in the map, the pipe will be colored using color 2.

To avoid overlapping colors, only use less than (<) as the last entry. A better map would be the following:

- >=60
- >=20
- <20

To delete a map value, select the row from the Color Map Grid and click the Remove Row button. Instead of coloring the pipes, it is possible to color just the pipe numbers. This is done by choosing the Color Pipe Labels radio button in the Color Options area. Alternatively, the Color Pipes and Labels radio button allows both to be colored.

Color choices from the Color Map Grid will appear in a legend without units shown by default. The legend may be repositioned by dragging it to any location on the [Workspace](#). Units may be displayed in the legend using the Show Units in Legend setting within Global Layer Settings.

If there is no output for the current scenario, any Color Map layer that displays an output parameter will have a red exclamation mark next to it on the Layers panel to warn the user that there is no output. The model must run before such Color Map layers can be displayed.

Save/Load Layers to File

There are two buttons on the Layers Toolbar that can be used to save or load Layer Template Files which enable teams to create consistent layer formatting across different models.

Save Layers to File - Specific layers can be selected, including Color Maps/Animations, which will be exported to a Layer Template File (*.lrf). You can optionally choose for Global Layer Settings to be saved in the template file. The All Objects Layer cannot be selected or saved.

Load Layers from File - A Layer Template File (*.lrf) can be imported which will add layers on top of any existing layers in the model and overwrite the Global Layer Settings if they were selected to be saved in the template. If the template file contains parameters that are not available then they will be ignored and not loaded. It is also possible to load old Visual Report Control Files created from previous versions of the software. Model files themselves can also be selected as a supported file type which will copy all available layers from the model.

Workspace Layers Import Assistant

The Workspace Layers Import Assistant can be used when importing a model from Fathom 12 or earlier. By default, when opening a model from these previous versions, layers are created as applicable for object numbers, workspace symbols, and object names. This is done regardless of whether or not the Workspace Layers Import Assistant is run.

The purpose of the Workspace Layers Import Assistant is to import visual styles of pipes and junctions and to import labels from the deprecated Visual Report Window as Workspace Layers so that previous Visual Reports can be recreated. This is a step by step guided procedure that will attempt to bring these elements into Workspace Layers while creating as few new layers as possible. These layers are created on a per-scenario basis.

If a model has a high number of scenarios with similar Visual Reports, it may be preferable to recreate the Visual Report manually as a new layer that can be used with any scenario. If the chosen scenarios for the Import Assistant share many of the same visual settings, the Import Assistant will create all relevant layers for each, even if those layers are identical to each other.

The Workspace Layers Import Assistant will allow you to select which scenarios to use to create the new layers. Once these are selected, the next page of the Import Assistant will allow you to see how many layers and Layer Presets will be generated. Once satisfied with the proposed Workspace Layers and Layer Presets, click Import Layers to complete the import process.

Related Topics

Workspace Window.....	384
Annotations.....	395
Merging Models.....	400

Renumbering Workspace Objects.....	399
Scale/Flip Workspace.....	401
Magnifier Tool.....	401
Extended Model Check.....	406
Keyboard Shortcuts for Workspace.....	404
Undefined Objects Window.....	403
Raw Data Changes.....	407
Duplicate Special.....	402

Annotations

Annotations are objects that can be placed on the [Workspace](#) to display customized text, express ideas via shapes or lines, or even to insert pictures into the Workspace. These are purely visual and do not affect results; their purpose is to communicate conceptual information about a system. They appear both in the Workspace and when printing from the Workspace.

New annotations are created using the Annotation Tool in the [Toolbox](#). The Annotation Tool is located to the right of the Pipe Drawing Tool. With this tool, the user can draw a line or a shape on the Workspace. Shapes can display an image or text on the Workspace or can be used without text to display boundaries around various regions of the model.

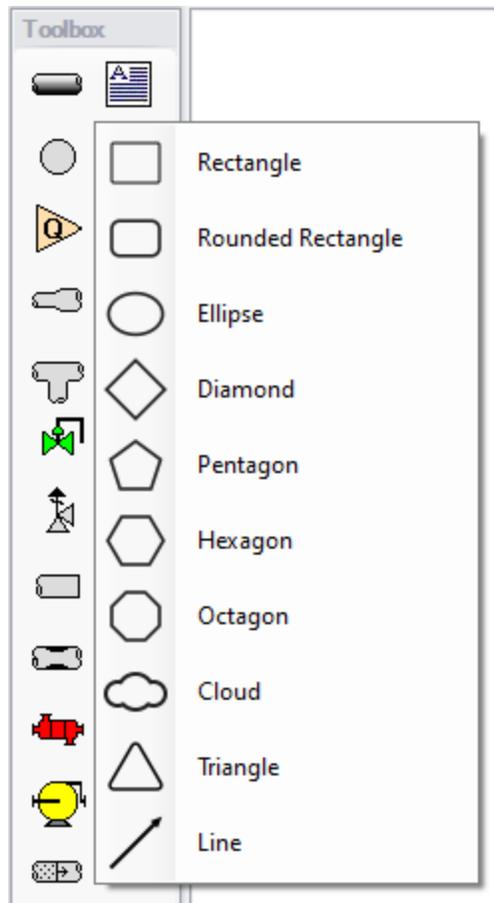


Figure 1: After clicking the Annotation Tool, select the shape for the new annotation

Properties

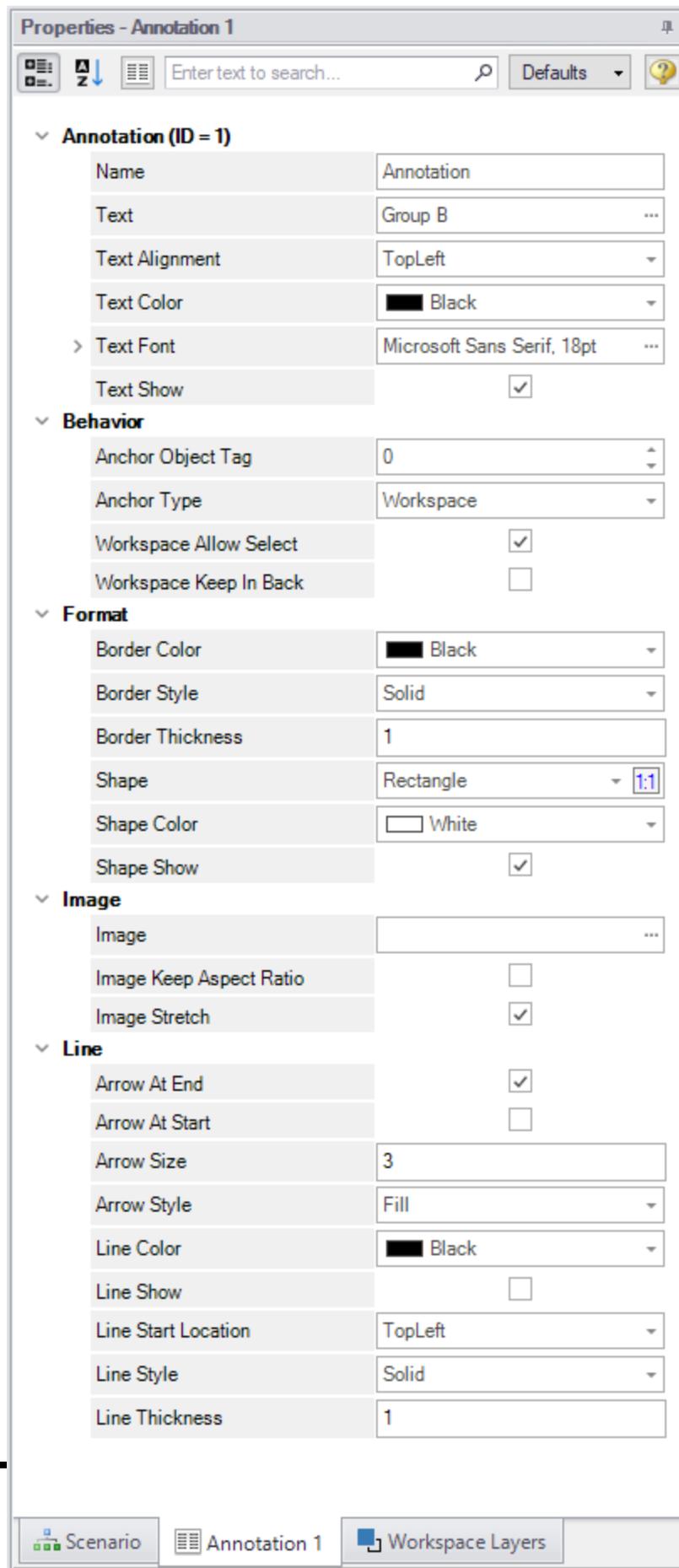


Figure 2: Annotation view for the Properties in the Quick Access Panel

After an annotation is placed on the Workspace, the properties of the annotation will be displayed in the Properties tab of the [Quick Access Panel](#) where they can be edited. For existing annotations, it can be accessed quickly by double-clicking on the annotation in the Workspace. Annotation styles can be set as default or restored to default settings using the Defaults drop-down menu in the upper right of the Annotation properties area.

- **Annotations** - The name of the annotation, its text content, text style, and text visibility can be edited within the Annotations section. The name of the annotation is not displayed on the Workspace but is shown within the [Layers Settings](#) where it can be used to communicate the purpose of an annotation.
- **Behavior** - The Behavior section allows the behavior of the annotation to be customized. Annotations can be attached to the Workspace, a pipe, or a junction. When attached to the Workspace, the annotation always remains in the same location until you move it. When attached to a pipe or junction, the annotation will move when the pipe or junction is moved.

Note: Whether an annotation appears behind or in front of another object is usually determined by the order of the Workspace Layers in which they are visible, but the toggle Workspace Keep in Back forces the annotation to appear below pipes and junctions regardless of layer order.

- **Format** - The Format section controls the shape as well as the shape's style and visibility.
- **Image** - The Image section allows an image to be displayed within the annotation and controls its appearance.
- **Line** - The Line section controls the appearance of lines and only applies when the shape is set to be a line in the Format section. Similarly, the shape color and border properties from the Format section are not used for lines.

Interaction with Scenarios and Layers

Annotations are not scenario-specific and exist globally within a model. Their visibility is associated with layers and can be controlled for each layer in the [Layer Settings](#). While annotations exist globally, their visibility for a given scenario can be controlled using [Layer Presets](#).

Related Topics

Workspace Window.....	384
Workspace Layers.....	385
Merging Models.....	400
Renumbering Workspace Objects.....	399
Scale/Flip Workspace.....	401
Magnifier Tool.....	401
Extended Model Check.....	406
Keyboard Shortcuts for Workspace.....	404
Undefined Objects Window.....	403

Raw Data Changes.....	407
Duplicate Special.....	402

Renumbering Workspace Objects

Objects on the [Workspace](#) can be renumbered by choosing Renumber from the Edit menu. There are several ways that Renumbering can proceed.

Renumber Wizard

The Renumber Wizard allows you to renumber pipes and/or junctions by simply clicking on them on the Workspace. Normal Workspace functions are suspended when the Renumber Wizard is active. The Renumber Wizard is activated on the [Edit menu](#).

When the Renumber Wizard is active the settings can be modified in the [Quick Access Panel](#). Click Done Renumbering in the Quick Access Panel to finish renumbering and exit the wizard.

Renumber Automatic

Selected [pipes and/or junctions](#) can be renumbered starting at the lowest selected number or at a specified number. If a pipe or junction which is not selected to be renumbered already has the next number in sequence, it will only be changed if the Force Sequential checkbox is checked.

Fathom will start at the beginning of the flow path containing the lowest numbered selected pipe and will attempt to march down the pipes in the direction of flow. It is recommended that renumbering be performed after the model has been completed so that the junctions and pipes are completely connected.

Tip: If you want certain sections of the model to be numbered in sequence, then use the Renumber feature on the selected sections only. It is also preferable in the case of large or complicated models to renumber small sections at a time.

Renumber Increment

Renumber Increment is on the Edit menu and allows you to increment or decrement all selected pipes and junctions by a specified value.

Related Topics

Workspace Window.....	384
Workspace Layers.....	385
Annotations.....	395
Merging Models.....	400
Scale/Flip Workspace.....	401
Magnifier Tool.....	401

Extended Model Check.....	406
Keyboard Shortcuts for Workspace.....	404
Undefined Objects Window.....	403
Raw Data Changes.....	407
Duplicate Special.....	402

Merging Models

Two models can be merged together using the command on the File menu. Here's how:

- Open the first model
- Choose Merge on the [File menu](#)
- From the file list, select the model to be merged

If there are duplicate numbers, the objects in the second model will be changed. The second model will be selected (highlighted) so that you can move it into position (be careful not to click off the selected portions or the second model may be deselected).

Note that annotations, layers, and layer presets are not imported from the second model.

Tip: To move a selected group of objects, use a junction icon, instead of a pipe, to drag the group. This will avoid missing the pipe and clicking on an empty area of the [Workspace](#) which will deselect your group.

Merging Models with Multiple Scenarios

If the model you are merging from has multiple scenarios, only the base scenario can be merged. If you want to merge one of the children of that model, do the following:

1. Open the model which you want to merge.
2. Open the [Scenario Manager](#).
3. Select the child scenario you want to merge.
4. On the Other Actions button, choose the Save Scenario to File Without Children.
5. Enter a name and click OK. This process creates a new model with the desired scenario as the base scenario.
6. Open the original model and merge in the new model file created in Step 5.

Related Topics

Workspace Window.....	384
Workspace Layers.....	385
Annotations.....	395
Renumbering Workspace Objects.....	399
Scale/Flip Workspace.....	401
Magnifier Tool.....	401

Extended Model Check.....	406
Keyboard Shortcuts for Workspace.....	404
Undefined Objects Window.....	403
Raw Data Changes.....	407
Duplicate Special.....	402

Scale/Flip Workspace

The [Workspace](#) can be scaled horizontally and/or vertically by choosing Scale/Flip Workspace on the [Arrange menu](#). Scaling will move the junctions and will stretch or shrink the pipes so that the area the model covers is changed by the percentage specified.

Flipping the Workspace horizontally and/or vertically will move the pipes and junctions to their mirror image locations. This is useful when you are building a system which has symmetrical or mirrored sections. For example, you could select the section, duplicate it, and then flip it horizontally, vertically, or both with Scale/Flip Selected Objects Only checked.

Related Topics

Workspace Window.....	384
Workspace Layers.....	385
Annotations.....	395
Merging Models.....	400
Renumbering Workspace Objects.....	399
Magnifier Tool.....	401
Extended Model Check.....	406
Keyboard Shortcuts for Workspace.....	404
Undefined Objects Window.....	403
Raw Data Changes.....	407
Duplicate Special.....	402

Magnifier Tool

The Magnifier tool can be used to provide a magnified view of any area on the Workspace. This is useful for navigating and viewing details of a large model without needing to zoom in on that area. The Magnifier tool can be accessed from the Workspace Toolbar, as can be seen in Figure 1 below. The magnifier tool can be enabled/disabled by clicking the button on the Toolbar.

There are several options available to adjust the Magnifier view:

- **Zoom Level** - Adjusts the level of magnification used
- **Magnifier Size** - Adjusts the size of the magnifier circle

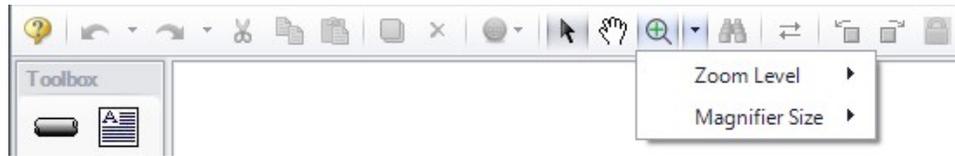


Figure 1: Magnifier tool on the Toolbar

Related Topics

Workspace Window.....	384
Workspace Layers.....	385
Annotations.....	395
Merging Models.....	400
Renumbering Workspace Objects.....	399
Scale/Flip Workspace.....	401
Extended Model Check.....	406
Keyboard Shortcuts for Workspace.....	404
Undefined Objects Window.....	403
Raw Data Changes.....	407
Duplicate Special.....	402

Duplicate Special

Duplicate Special allows the model to be copy & pasted, while simultaneously applying other helpful settings as explained below. Additional options will be available when the [ANS module](#) is activated.

- **Increment All Pipe and Junction Numbers By** - Adds the specified number to all pipe/junction numbers for the created object duplicates. Useful for maintaining consistent numbering
- **Create Group** - Adds the duplicates to a [group](#).
- **Hide in Output** - Automatically hides the pipe and junction duplicates in the Output window so only the original pipes and junctions will be seen.
- **Copy Layer Style Settings From Original Objects** - Duplicates [Workspace Layers](#) style settings (visibility, thickness, color, label parameters, etc.) for selected objects.
- **Make Dependent Design Case** - (ANS Only) - Sets the duplicate objects as a [Dependent Design Case](#) of the duplicated model.
- **Keep Common Size Groups** - (ANS Only) - Only available if "Make Dependent Design Case" is checked. This adds the duplicate objects to the same [Common Size Groups](#) that the original objects were included in.
- **Clear Design Requirements** - (ANS Only) - This clears all [Design Requirements](#) from the newly created objects.
- **Link GSC Variables in New Pipes/Jcts** - (XTS Only) - Variables associated with duplicated objects are duplicated and linked together within the [Variables Panel](#).
- **Duplicate Goals** - (XTS Only) - Goals associated with duplicated objects are duplicated.

Related Topics

Workspace Window.....	384
Workspace Layers.....	385
Annotations.....	395
Merging Models.....	400
Renumbering Workspace Objects.....	399
Scale/Flip Workspace.....	401
Magnifier Tool.....	401
Extended Model Check.....	406
Keyboard Shortcuts for Workspace.....	404
Undefined Objects Window.....	403
Raw Data Changes.....	407

Undefined Objects Window

This window is launched by clicking List Undefined Objects from the [View menu](#).

This window displays all undefined pipes or junctions in the model. Clicking on the undefined object will display the undefined properties in the list below. All undefined data for all objects can be saved to the file or copied to the clipboard.

Any undefined object's Properties window can be launched by double-clicking the object's name in the list to finish defining the object.

Related Topics

Workspace Window.....	384
Workspace Layers.....	385
Annotations.....	395
Merging Models.....	400
Renumbering Workspace Objects.....	399
Scale/Flip Workspace.....	401
Magnifier Tool.....	401
Extended Model Check.....	406
Keyboard Shortcuts for Workspace.....	404
Raw Data Changes.....	407
Duplicate Special.....	402

Keyboard Shortcuts for Workspace

Selection Box

- Select all objects completely inside box by dragging cursor from left to right
- Select all object completely or partially inside box by dragging cursor from right to left

Pipe Drawing Tool

- If you double-click the pipe drawing tool, it will allow you to draw multiple pipes at a time. Clicking Esc or clicking the pipe drawing tool again will deactivate the tool.
- When [orthogonal drawing](#) is off and you hold down the SHIFT key while drawing pipes on the Workspace, you can [draw pipes](#) that are perfectly horizontal or vertical. When the orthogonal drawing is on and you hold down the SHIFT key down while drawing new pipes on the Workspace the pipe vertex will flip. If you move an existing pipe and hold the shift key while selecting a handle at one of the pipe endpoints a new vertex will be inserted at that point endpoint.
- If you hold down the CTRL key when completing the pipe drawing (just before releasing the mouse button), the Pipe Drawing Tool remains active, and you can draw a series of pipes without returning to the Toolbox each time.
- To add [pipe segments](#), use CTRL+SHIFT+Y
- To remove all [pipe segments](#), use CTRL+SHIFT+N

Junctions

- If you double-click any junction in the Toolbox, you will be able to place multiple of that junction on the Workspace by clicking the Workspace where you would like to place each junction. Clicking Esc or clicking the junction icon in the Toolbox again will deactivate the tool.
- If you hold down CTRL key and select a [junction](#) from the Toolbox and drop it onto an existing Workspace junction, the old junction will "[morph](#)" into the new one. The junction type will change to the new junction type, and data that can be kept will be copied into the new junction.
- If you hold down the CTRL key then drag a junction on the Workspace which has pipes connected to it, the junction will move but the pipes will not move with it
- If you hold down the SHIFT key while selecting a junction from the Toolbox and then drop the junction onto an existing pipe, the pipe will split into two pipes. You will specify the length of the first pipe and the second pipe will be automatically set to the remaining length from the original pipe. Thus the sum of the two pipe lengths will equal the length of the original pipe. You can also choose which pipe contains any Fittings & Losses that were specified within the original pipe.

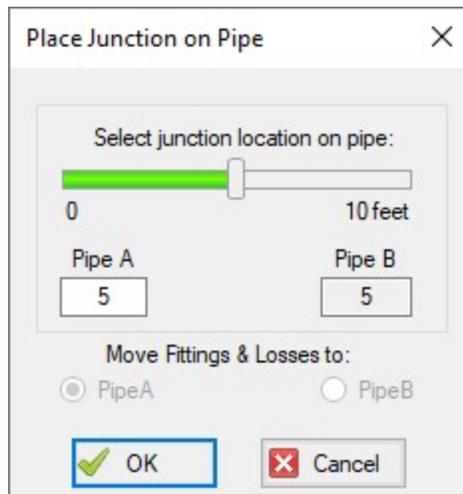


Figure 1: Split pipe keyboard shortcut

Pipe and Junction Properties Window

- When in a pipe or junction Properties Window you can toggle the highlight feature by pressing the F2 function key. You can also toggle this from the View menu.
- When in a [Pipe Properties window](#), if you press the F5 function key, you will jump to the next highest numbered pipe. If you press the CTRL key while pressing the F5 function key, you will jump to the next lowest numbered pipe. The F5 function key also behaves this way for junction Properties Windows.

Pan and Zoom

- The Workspace position can be moved with CTRL + arrow keys.
- The Zoom level can be adjusted with CTRL + (+/-)
- The Zoom level can be reset to default with CTRL + 0
- Mouses with scroll wheels can be used to zoom by scrolling, or to pan by clicking and holding the scroll wheel

Switching Primary Windows

- Go to the Workspace window using CTRL + W
- Go to the Model Data window using CTRL + M
- Go to the Output window using CTRL + U
- Go to the Graph Results window using CTRL + G
- When [ANS](#) is active, go to the Sizing window using CTRL + N

Other

- Open the Keyboard Shortcuts Help Page (this page!): CTRL + Q
- Run the model: CTRL + R
- Save the model: CTRL + S
- Open a file: CTRL + O
- Undo the last action: CTRL + Z
- Redo the last action: CTRL + Y
- Cut: CTRL + X
- Copy: CTRL + C
- Paste: CTRL + V
- Duplicate: CTRL + D
- Delete selected pipes and junctions: DEL
- Reverse pipe direction: F3
- Lock Object: CTRL + L
- Toggle special condition of pipes/junctions between open and closed: F4
- Show object status: CTRL + H
- Other available shortcuts can be seen listed by their command on the menus

Related Topics

Toolbox.....	195
Creating Pipes - the Pipe Drawing Tool.....	197
Creating Junctions.....	202
Workspace Window.....	384
Workspace Layers.....	385
Annotations.....	395
Merging Models.....	400
Renumbering Workspace Objects.....	399
Scale/Flip Workspace.....	401
Magnifier Tool.....	401
Extended Model Check.....	406
Undefined Objects Window.....	403
Raw Data Changes.....	407
Duplicate Special.....	402

Extended Model Check

The connections and settings of each pipe and junction are exhaustively checked using the Extended Model Check item on the [View menu](#). This is also helpful if the model seems to not be connected correctly. Note that this clears Output for the current Scenario.

Related Topics

Workspace Window.....	384
Workspace Layers.....	385
Annotations.....	395
Merging Models.....	400
Renumbering Workspace Objects.....	399
Scale/Flip Workspace.....	401
Magnifier Tool.....	401
Keyboard Shortcuts for Workspace.....	404
Undefined Objects Window.....	403
Raw Data Changes.....	407
Duplicate Special.....	402

Raw Data Changes

When other objects in the model use the same base raw data as the current object and the data is changed, you can decide whether to use the new data on the other model objects.

This frequently happens when you have multiple junctions based on the same x-y data for pump curves or junctions resistance curves, and later change that x-y data. The question is whether you want to have the current junction you are working on updated, or all junctions that use that same base data.

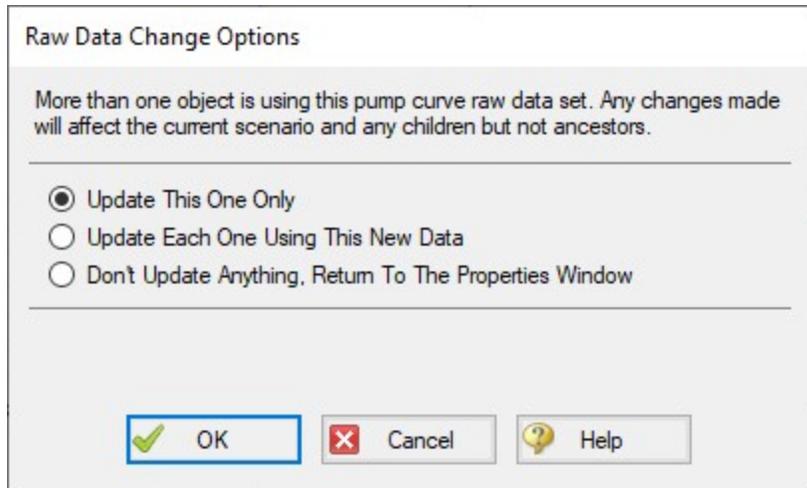


Figure 1: Raw Data Change Options

Related Topics

Workspace Window.....	384
Workspace Layers.....	385
Annotations.....	395

Merging Models.....400
 Renumbering Workspace Objects.....399
 Scale/Flip Workspace.....401
 Magnifier Tool.....401
 Extended Model Check.....406
 Keyboard Shortcuts for Workspace.....404
 Undefined Objects Window.....403
 Duplicate Special.....402

Model Data Window

The Model Data window offers a text-based perspective on all engineering information in the pipe flow model. This window is immensely useful for obtaining a global view of the model and rapidly checking the input.

The Model Data window does not offer any tools to build or add to a model; all model assembly must happen in the [Workspace](#).

The Model Data window allows you to manipulate *existing* elements from the Workspace. Therefore, a complete model can be assembled in the Workspace window without ever opening the Model Data window. In fact, for small models this may be the preferred approach.

General, Pipe, and Junction section display sizes can be changed by the user by clicking and dragging the bars between each section to resize them. Each section can also be hidden by clicking the arrow in the upper left of each section.

You can use the zoom feature in the lower right hand corner of the Model Data window to resize the text displayed on screen.

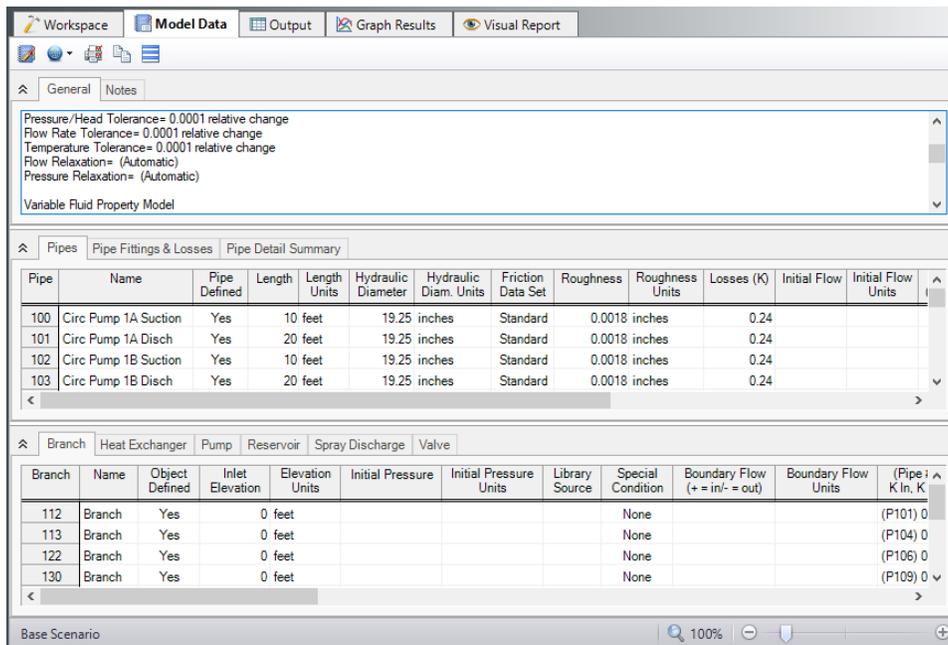


Figure 1: The Model Data window displays input in text form

Editing Features

The Pipe table allows display of all input data for all pipes in the model. Details of [Fittings & Losses](#) are displayed in the Pipe Loss Data table, accessed with an adjacent tab. The Pipe Loss Data table displays all fittings with K factors grouped into columns.

The other display tab in the pipe area is the Pipe Detail Summary, which assembles a list of details about each pipe in another format useful to some engineers.

The junction data area is below the pipe data area. Here the junction data is separated into tables for each junction type. Click the tabs to see the data for any type of junction.

To edit information in the Model Data window, you can open the Properties Window for the pipe or junction. This can be done by double-clicking anywhere in the row for the pipe or junction to be edited. Alternatively, you can use the [Global Edit](#) windows opened from the Edit menu.

- **XTS Module tables** - When using the [XTS Module](#), a Transient Data table in the Junction area displays all transient data entered for junctions. If there is no transient data, the table will not be displayed.
- **GSC Module tables** - When using the [GSC Module](#), if GSC data has been entered in the [Goal Seek and Control Group](#) in [Analysis Setup](#), the variable and goal data will be displayed in the General section in a "Goal Seek & Control" table.

Display and Printing Features

[Model Data Control](#) lets you specify the combination of pipe, junction, and general information you want to see in the Model Data Window.

The Model Data Print Content window lets you specify the content to include in the printed Model Data report and select the font to be used for printing. This window is opened from the [Tools menu](#) (or [Toolbar](#)).

The Model Data window contents can be:

- Copied to the clipboard
- Saved to a formatted text file
- Exported to a delimited file suitable for direct importing into spreadsheet software
- Saved directly to an Adobe PDF file
- Sent to the printer

Related Topics

Interface and User Options.....	359
Menus and Toolbars.....	359
Model Data Control.....	410
Bill of Materials.....	412

Related Examples

Beginner - Three Reservoir823

Model Data Control

The Model Data Control allows you to customize what data to show for the pipes and for each junction type and also which pipes and junctions to show (see Figure 1). It can be opened from the [Tools menu](#) (or Toolbar).

Pipe Data Display

The data to be shown for the pipes is selected on the left side of the Model Data Control window. Only the items with check marks will be shown in the Model Data window. All pipes shown will report the selected items.

Junction Data Display

The data to be shown for the junctions is selected on the right side of the Model Data Control window. The data is grouped by junction type. However, some data is general and applied to each of the junction types. Select the junction type from the list and then select the data for that junction type. Each type also includes the general data which may be selected/deselected.

You can select/deselect the general data for all junction types simultaneously by clicking Set Common.

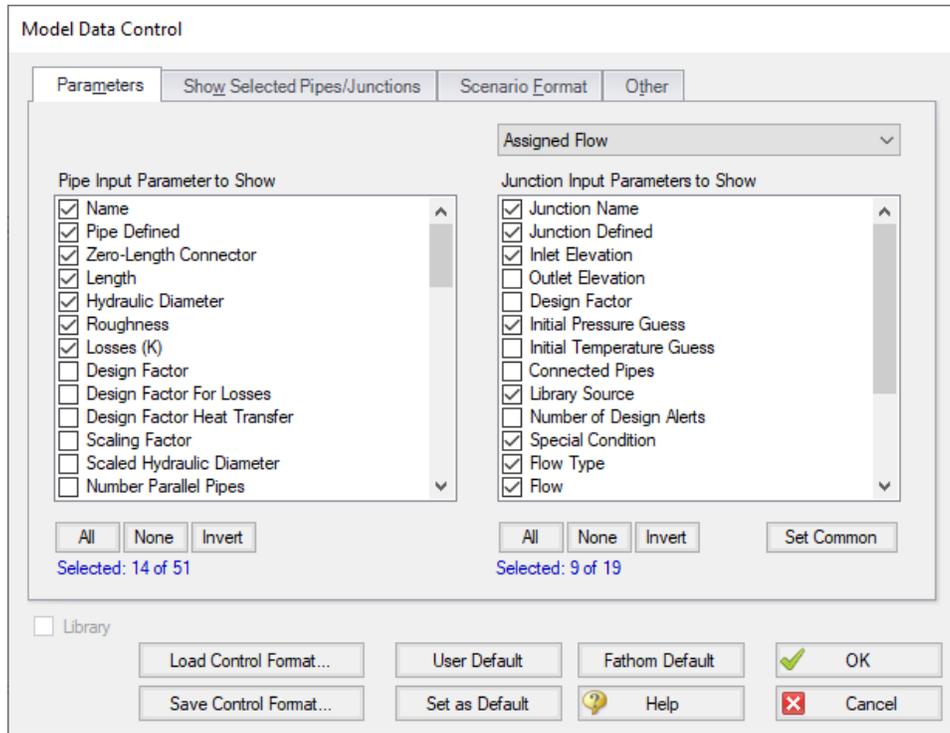


Figure 1: The Model Data Control window allows selection of the parameters to show in the Model Data window

Model Data Format

The Model Data Control format can be saved to a file so that the format can be quickly applied to other model files. There are multiple ways to do this. The Model Data Control format can be saved to the internal user library by clicking the Set As Default button, then choosing User Default in a new model file to apply the settings.

The Model Data format can also be saved to an external file, which can then be shared with different users. The external file can be generated by clicking the Save Control Format button at the bottom of the window. The format can then be applied to other model files by clicking the Load Control Format button.

Show Selected Pipes and Junctions

Specific pipes and junctions can be shown in Model Data. Click the Show Selected Pipes/Jct tab, then click the items in the lists to select the pipe(s) and junction(s) to be shown.

Other Data

When pipes with intermediate elevations are specified, you can specify which pipe data is to be displayed in the Model Data window:

- Show only pipes with intermediate elevations
- Show only non-horizontal pipes
- Show all pipes

In addition, you can specify whether to show [Pipe Elevation Data](#), whether junction curve data is to be displayed with the Model Data, whether to display [Design Alerts](#), and whether to include a [Bill of Materials](#).

Scenario Format

A powerful Model Data feature when using scenarios is to show all direct ancestor data in the Model Data along with data from the current scenario. The Scenario Format features allow you to do this, and to highlight where data changes occur. Note that the scenario name is appended to the pipe and junction number in the left columns. This tab only appears when multiple scenarios exist.

Library Connections

In the lower left of the Model Data Control window there is a Library checkbox. If this box is checked, your Model Data Control parameters are setup as determined by the library to which you are [connected](#). This is referred to as an *active* library. To make it inactive, uncheck the box or change one of the Model Data Control settings controlled by the library.

If the check box is unchecked, but enabled, you are connected to a library but the settings are not being passed to the Model Data Control window. The library is thus *inactive*. To make it active, check the box then click the OK button.

If the check box is disabled, there is no connected library.

Related Topics

Model Data Window.....	408
Bill of Materials.....	412

Bill of Materials

A bill of materials is a summary of all materials, components, and fittings that have been specified for a project.

A bill of materials can be added to the Model Data Summary section by opening Model Data Control, and checking "Include Bill of Materials" on the Other tab.

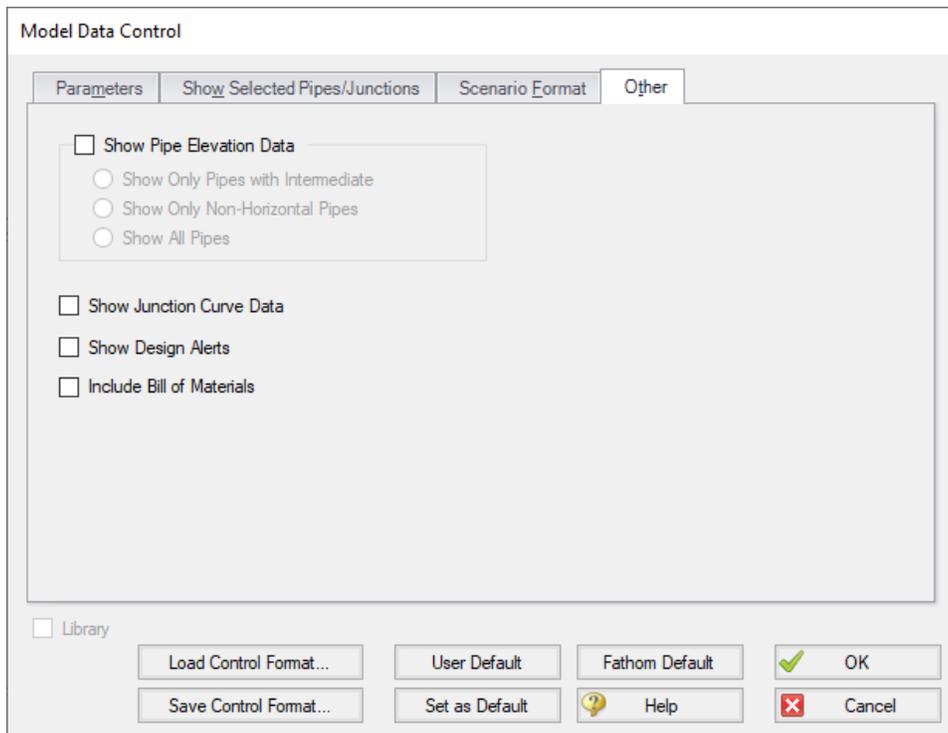


Figure 1: Include Bill of Materials

Related Topics

Model Data Window.....	408
Model Data Control.....	410

Output Window

The Output window is the primary vehicle for communicating the results of an analysis in text form (see Figure 1). The Output window follows the same general window organization as the Model Data window.

When a model is submitted to the Solver (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system), the [Solution Progress window](#) appears. After the solution is obtained, you can view the results in the Output window by clicking the View Output button.

If the information in the Output window is not in the desired format, you can choose [Output Control](#) from the Tools menu (or [Toolbar](#)) to modify the format. Output Control lets you specify the parameters, their units, and the order of their display in the Output window.

The Output Print Content window available on the Tools menu (or [Toolbar](#)) lets you specify the content to include in the printed Output report and select the font to be used.

The **Sort** window opened from the Arrange menu (or [Toolbar](#)) offers the ability to sort the Pipe Results or Junction Results tables according to the values in any of the columns.

The [Transfer Results to Initial Guesses](#) feature on the Edit menu (and Toolbar) sets all pipe and junction initial flows and pressures equal to the converged results, which causes future runs to converge much faster.

You can view the input data for individual pipes or junctions directly from the Output window. To do this double-click the pipe or junction row, which will open the properties window in read-only form where you can review input in detail.

General, Pipe, and Junction section display sizes can be changed by the user by clicking and dragging the bars between each section to resize them. Each section can also be hidden by clicking the arrow in the upper left of each section.

You can use the zoom feature in the lower right hand corner of the Output window to resize the text displayed on screen.

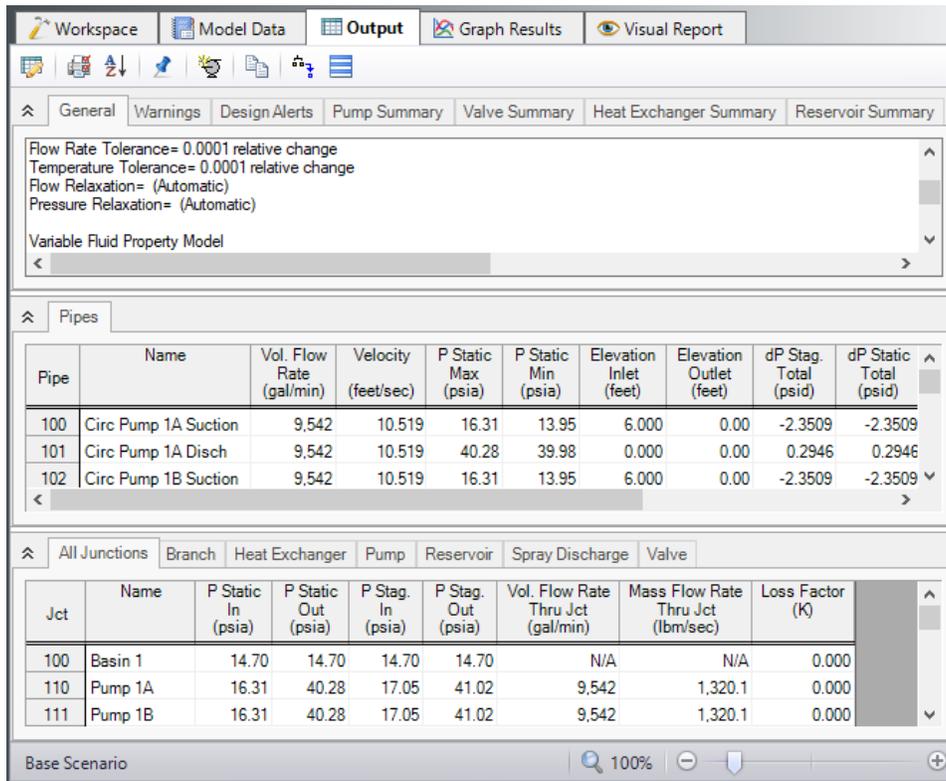


Figure 1: The Output Window

Right-click Context Menu Actions

There are several options available when right-clicking on a cell, row header, column header, or entire table in the Output window:

- **Column Header**
 - **Change Units** - Quickly redisplay the selected column in a different unit.
 - **Sort** - Sort the table in Ascending or Descending order on the selected column.
 - **Add "Parameter" for all Table Objects to Excel Export Manager** - Add this column to the [Excel Export Manager](#).
 - **Open Excel Export Manager** - Directly open the [Excel Export Manager](#).
 - **Copy Selected Cell Data** - Copies all of the values in the column to the clipboard.
- **Row Header**
 - **Add all Object Parameters to Excel Export Manager** - Add this row to the [Excel Export Manager](#).
 - **Open Excel Export Manager** - Directly open the [Excel Export Manager](#).
 - **Find on Workspace** - [Quickly locate](#) an object on the Workspace.
 - **Copy Selected Cell Data** - Copies all of the values in the row to the clipboard.
 - **Open Properties Window** - Opens the properties window for the associated pipe or junction.
- **Table (upper left cell)**
 - **Add Object Table to Excel Export Manager** - Add the entire table to the [Excel Export Manager](#).
 - **Open Excel Export Manager** - Directly open the [Excel Export Manager](#).
 - **Copy Selected Cell Data** - Copies all of the values in the table to the clipboard.
- **Cell**
 - **Add Object "Parameter" to Excel Export Manager** - Add this cell to the [Excel Export Manager](#).
 - **Open Excel Export Manager** - Directly open the [Excel Export Manager](#).
 - **Create Design Alert** - Create a [Design Alert](#) related to the selected object and value.
 - **Copy Selected Cell Data** - Copies the value of the cell to the clipboard.

Output Window Updates

Whenever you change the input model, all output windows are erased. This prevents you from changing your input model, forgetting to rerun it, and mistaking the previous Output window for the current model's results.

Results Diagrams

Results diagrams are available for both pump results (see Figure 2) and pipe heat transfer results (see Figure 3). These diagrams allow you display the information in a more intuitive manner, copy the image of the diagram to your clipboard, or print the diagram. The option to show the results diagram can be found in the pump summary tab for pumps or the heat transfer tab for pipes with heat transfer.

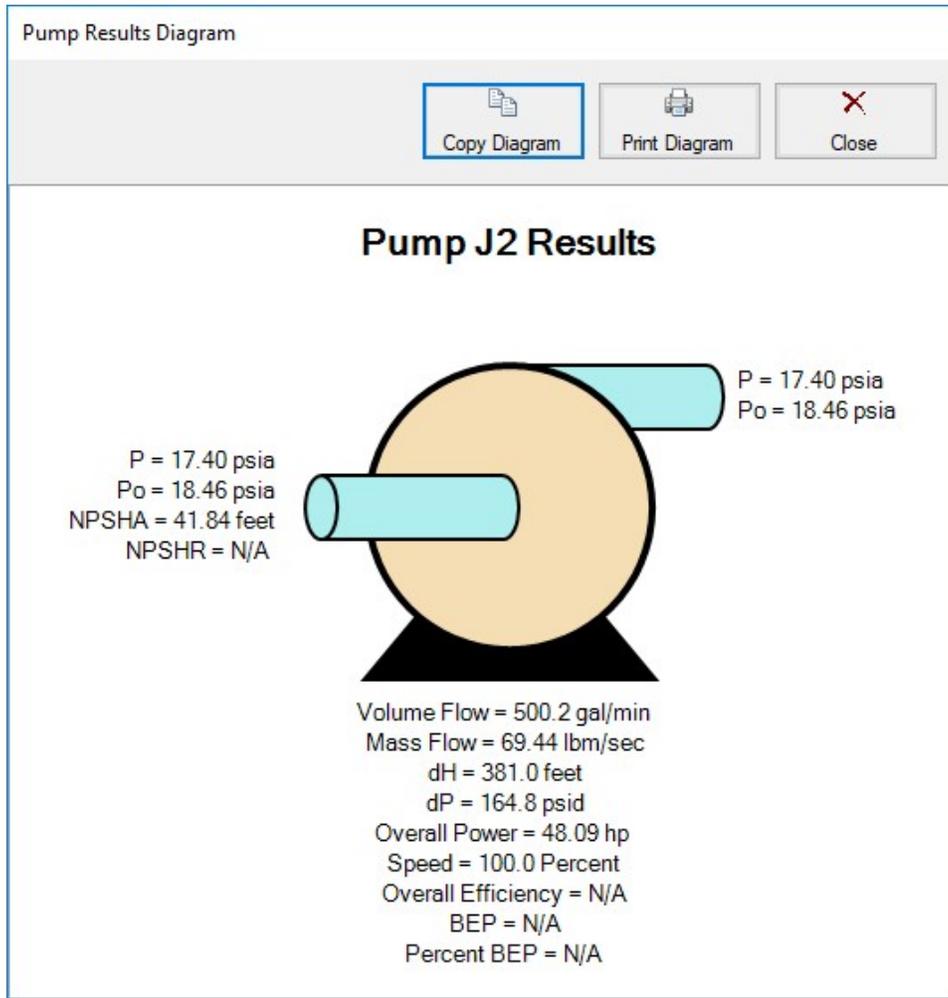


Figure 2: Pump Results Diagrams

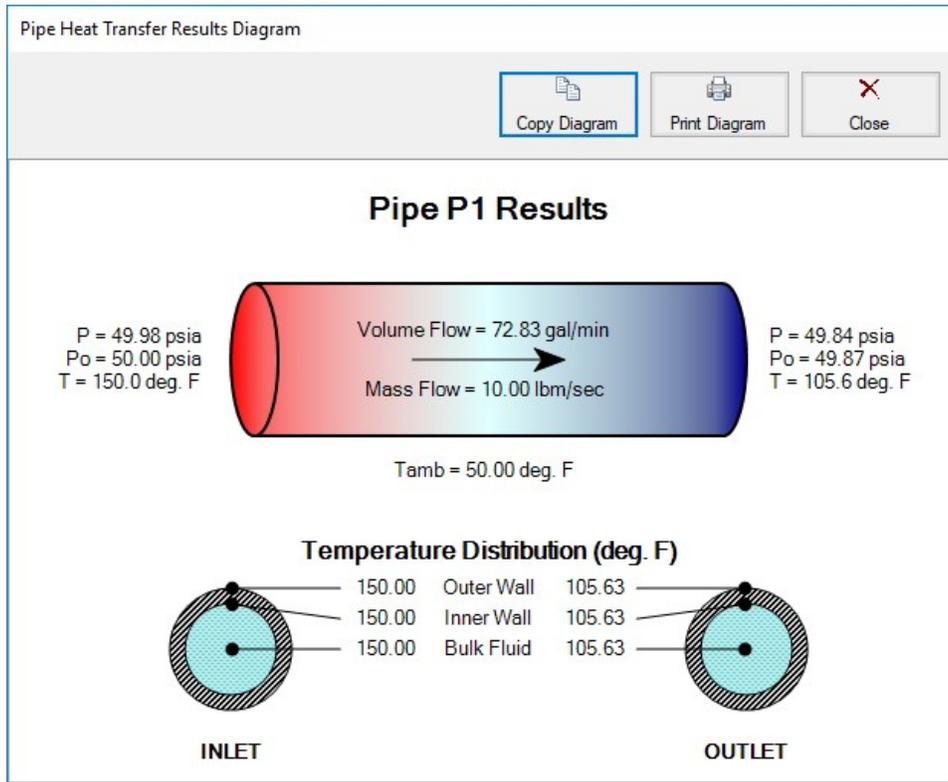


Figure 3: Pipe Heat Transfer Results Diagram

Printing and Exporting

Data in the Output window can be:

- Copied to the clipboard
- Saved to a formatted text file
- Exported to an Excel spreadsheet
- Saved directly to an Adobe PDF file
- Sent to the printer (see the [Printing](#) topic)

The printed and Adobe PDF output follows the same format as the Output window itself.

Related Topics

Interface and User Options.....	359
Menus and Toolbars.....	359
Output Control.....	418
Output Parameters.....	424

Applied Standards.....426
 Use the Output Window Sort Feature.....802
 Exporting To Excel.....472
 XTS in Output.....612
 GSC in Output.....543
 Cost Report.....535
 SSL in Output.....580
 Output Messages & Warnings.....772

Related Examples

Beginner - Three Reservoir823

Output Control

The Output Control window (see Figure 1) allows you to customize the parameters shown on the Output window. The control features are grouped on numerous tabs.

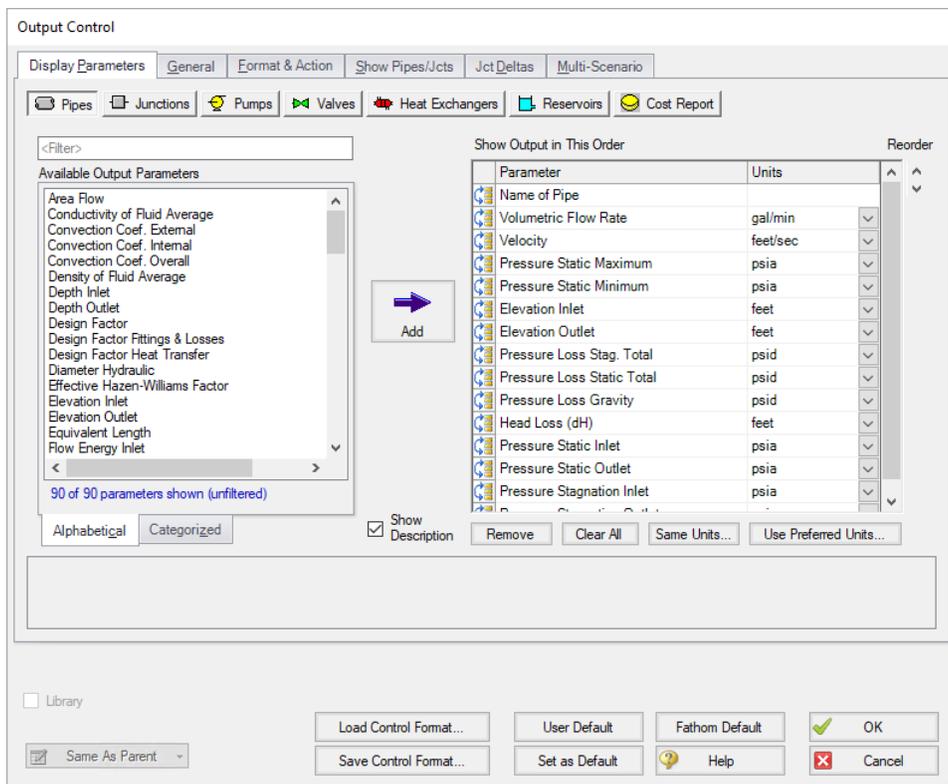


Figure 1: The Output Control window selects parameters for the Output window. Here the Pipe parameters are shown. The Junction and other parameters function the same.

Display Parameters

The Display Parameters tab controls the parameters to be shown for the pipes, junctions and summary reports. The output type is selected from the group of buttons at the top. For each output type, the list on the left side shows all available parameters for the selected report. For the pipe and junction output, this list can be viewed in alphabetized or categorized mode, which makes it easier to find a parameter.

Choose a parameter to be shown by selecting it on the left-hand list and clicking the Add button, or double-clicking the parameter name. The selected item will appear in the list on the right either at the bottom or at the previously selected location. The list on the right shows the order of the parameters to be displayed in the output and the units (if any) in which to display the results.

You can change the order of the output by selecting a parameter on the right-hand list, then clicking the Reorder scroll bar on the far right or by clicking and dragging the row using the handle on the left. The list will be reordered accordingly. The output is presented from left to right in the order defined in this list.

The units used for a parameter can be changed by selecting the parameter then choosing the desired units from the adjacent dropdown list. By selecting the units of interest, you can obtain your results in whatever units you find most convenient and meaningful. This means that you can enter all your input parameters in one set of units (or a variety of units, for that matter), and have all your output parameters in a completely different set of units.

To set selected units for both pipes, junctions and summary reports to the preferred units (set in the [User Options](#) window), click the Use Preferred Units button. If you change the units of a parameter and you would like that unit to be used for all similar parameters in the list or entire output, select the Same Units button.

You will note that some output parameters do not offer the ability to specify output units. These parameters either do not have units associated with them (such as Reynolds Number or friction factor), or are input parameters from a library (such as Pipe Nominal Size).

Several Special Summary Reports can be displayed in the Output window. These include the following:

- **Pump Summary**
- **Valve Summary**
- **Heat Exchanger Summary**
- **Reservoir Summary**
- **Pipe Heat Transfer Output**
- **Cost Report**

In addition, if you are using the XTS module there will be transient output reports available for pipes, junctions and many of the summary reports. If you are using the SSL module, there will be a slurry report available and transient slurry report if XTS is also active. If the ANS module is active the Pipe Sizing tab will be available, as well as Design Requirements and Library tabs for each pipe/junction which has design requirements or libraries assigned to them.

General

The General Section is the text area located at the top of the Output window. The project title, reference information and solution balance display are controlled here.

You can enter a descriptive Analysis Title which will be used in the [Model Data](#), [Output](#) and [Workspace](#) windows. A title is required and can have up to 100 characters.

You can keep a lengthy explanation or any other documentation about your model in the Reference Information section. Names of projects, individuals, and assumptions can all be kept with the model. This information can then be included in your reports to improve tracking.

There is an option "[Show Solution Balance Summary](#)" that is defaulted to off. This option toggles the display of a general output tab showing the total flow entering and exiting each junction, as well as the sum of those flows. This can be helpful in verifying the results as well as in some troubleshooting cases.

Format & Action

The results can be automatically sent to the printer or saved to a file when the model has reached convergence. Additionally, the results can be automatically saved as initial conditions when the model is run. By saving the results as initial guesses, the iteration time for AFT Fathom to arrive at a converged solution is reduced. The initial guess can also be transferred manually by selecting Transfer Results to Initial Guesses on the Edit menu or Toolbar. The model can also be automatically saved after the [initial conditions are transferred](#). The valve state (open, closed or failed) can also be saved along with the initial conditions.

When enabled, the Wrap Table Column headers option makes the output tables wrap the column headers an extra line if the header is wider than usual.

There are four formatting methods for the pipe and junction output data:

- **Always Align Decimal Point on Output** - allows magnitude of the data to be easily compared as the data is read down the column. However, a number whose magnitude is small may force larger numbers to show many digits. For example, assume that the minimum number of digits is set to four, and there are two output values: 1234.5 and .09876. These two values will be displayed in the output as:
 - 1234.50000
 - 0.09876
- **Show Only Number of Digits (not aligned)** - forces the data to use exactly the number of digits specified, which may result in the decimal point location changing from value to value. Using this method with the above example and four digits, the output will be displayed as:
 - 1235
 - 9.876E-02
- **Align Only When Max/Min < Limit** - aligns the data on the decimal point only if the maximum value in the column divided by the minimum value is less than a limiting order of magnitude that you set. If this ratio is greater than the limit, then each value in the column will be exactly the number of digits specified. Using the above example with a limit set to 4 then $1234.5/.09876 = 12500$

or an order of magnitude of 5. This is greater than the limit. The entire column will not be aligned and will be displayed as:

- 1235
- 9.876E-02
- **Use Exponential When Max/Min > Limit** - similar to the above method except that if the ratio is greater than the limit then the entire column will be shown in exponential format. Using the above example, the column would be shown as
 - 1.235E+03
 - 9.876E-02

These formatting options are for the entire Output window; however, the last two are calculated and applied on a column basis. In other words, if you choose the third option one column may be aligned but another may not be.

The output can be forced to always display in exponential notation by choosing the checkbox.

Options to adjust the colors for the headers and design alert highlighting are also available.

Cost Report Formatting

The [Cost Report](#) monetary numerical format can be specified.

Time Simulation Formatting

([XTS Module](#) only) XTS output can be displayed in different units of time. In addition, when running transients over hours or days there are formats which display time in formats such as dd:hh:mm:ss.

Show Pipes/Jcts

You can choose which pipes and junctions are to be displayed in the Output window. Clicking a selected item in the list on the Show Pipes/Jcts tab will deselect it. Clicking a deselected item will select it. This feature is especially useful when you are interested only in a certain area of the model.

You can quickly define the output you want by doing the following:

1. Select the pipes and junctions on the Workspace
2. Open the Output Control window and select the Show Pipes/Jcts tab
3. Click the Workspace button for both the pipes and junctions
4. Click the Show button

Junction Deltas

You can have AFT Fathom calculate the difference between two junctions for a particular parameter. These results are displayed in the General Section. A list of Junction Deltas is created by selecting the

upstream and downstream junctions, the parameter and units of interest, then adding it to the list by clicking the Add >> button. Some parameters will require that either the inlet or outlet value for the junction be specified. The list will contain the parameter symbol and the two junction numbers with "in" or "out" in parentheses. Different parameters can be used for each item in the list so you can, for example, choose pressure difference between junctions 2 and 10 and also temperature difference between junctions 45 and 16.

Multi-Scenario

You can view output from multiple [scenarios](#) concurrently in the Output window. To do this, select the Multi-Scenario tab. This tab is only visible when at least one child scenario exists. From this tab, choose which scenarios you would like to view concurrently by clicking on the Select Scenarios button located at the top left of the Multi-Scenario tab. The color of each scenario is automatically generated and can be regenerated by selecting "Reset All Colors". You can manually select the background color for the Output data of a given scenario by clicking on the ... button located in the row corresponding to the scenario.

You can also order the scenarios to appear in the Output window according to your preferences. To do this, arrange the scenarios in the order you would like them to appear by using the turquoise arrows at the bottom left corner of the Scenario Order and Colors box. Then, select how you would like the current scenario to be displayed by clicking the appropriate choice under the Display Current Scenario section. Figure 3 shows the Output window with multiple scenarios displayed.

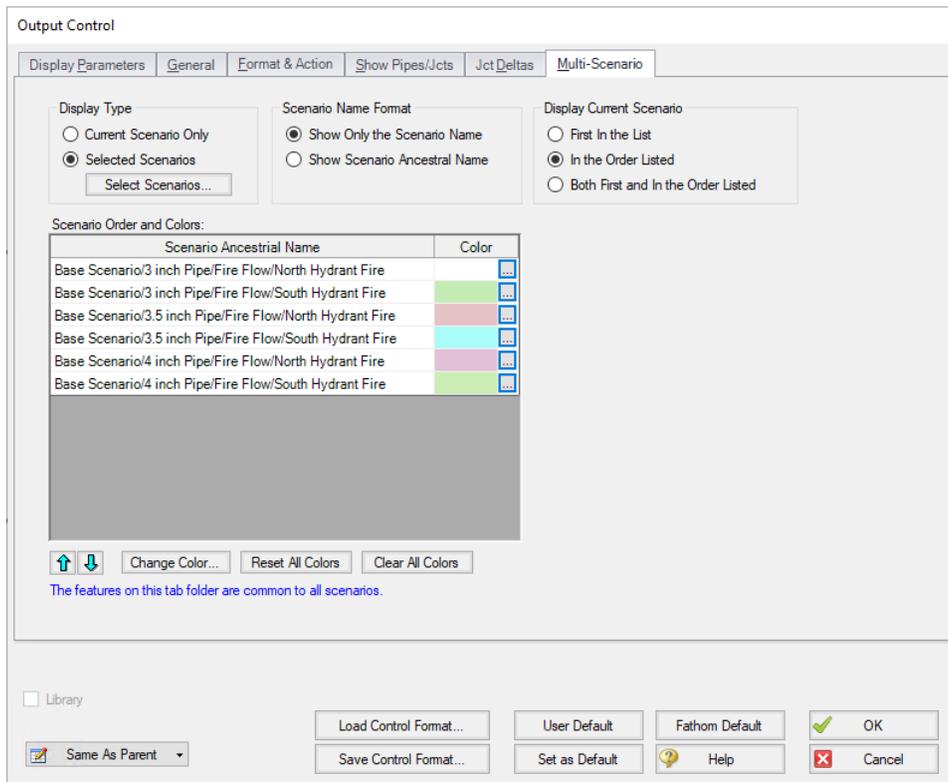


Figure 2: Output Control window selections for multi-scenario output

Pipe	Name	Vol. Flow Rate (gal/min)	Velocity (feet/sec)	P Static Max (psia)	P Static Min (psia)	Elevation Inlet (feet)	Elevation Outlet (feet)	dP Stag. Total (psid)	dP Static Total (psid)	dP Gravity (psid)	dH (feet)	P Static In (psia)	P Static Out (psia)	P Stag. In (psia)	P Stag. Out (psia)
1 - North Hydrant Fire	Pipe	120.000	5.2079	90.99	14.51	235.000	0.000	-76.47659	-76.47659	-101.855	58.55386	14.51	90.99	14.70	91.17
1 - South Hydrant Fire	N/A	120.000	5.2079	90.99	14.51	235.000	0.000	-76.47659	-76.47659	-101.855	58.55386	14.51	90.99	14.70	91.17
1 - North Hydrant Fire	N/A	120.000	3.8941	103.82	14.59	235.000	0.000	-89.22669	-89.22669	-101.855	29.13692	14.59	103.82	14.70	103.92
1 - South Hydrant Fire	N/A	120.000	3.8941	103.82	14.59	235.000	0.000	-89.22669	-89.22669	-101.855	29.13692	14.59	103.82	14.70	103.92
1 - North Hydrant Fire	N/A	120.000	3.0243	109.60	14.63	235.000	0.000	-94.96785	-94.96785	-101.855	15.89097	14.63	109.60	14.70	109.66
1 - South Hydrant Fire	N/A	120.000	3.0243	109.60	14.63	235.000	0.000	-94.96785	-94.96785	-101.855	15.89097	14.63	109.60	14.70	109.66
10 - North Hydrant Fire	Pipe	43.253	1.8771	91.15	88.37	0.000	5.000	2.78238	2.78238	2.167	1.41949	91.15	88.37	91.17	88.39
10 - South Hydrant Fire	N/A	71.584	3.1067	91.11	87.43	0.000	5.000	3.67544	3.67544	2.167	3.47995	91.11	87.43	91.17	87.50
10 - North Hydrant Fire	N/A	43.217	1.4024	103.91	101.44	0.000	5.000	2.47405	2.47405	2.167	0.70811	103.91	101.44	103.92	101.45
10 - South Hydrant Fire	N/A	71.614	2.3239	103.89	100.97	0.000	5.000	2.91977	2.91977	2.167	1.73646	103.89	100.97	103.92	101.00
10 - North Hydrant Fire	N/A	43.186	1.0884	109.66	107.32	0.000	5.000	2.33488	2.33488	2.167	0.38701	109.66	107.32	109.66	107.33
10 - South Hydrant Fire	N/A	71.641	1.8055	109.64	107.06	0.000	5.000	2.57858	2.57858	2.167	0.94927	109.64	107.06	109.66	107.09
20 - North Hydrant Fire	Pipe	-33.253	-1.4432	88.49	86.71	0.000	5.000	1.78087	1.78087	2.167	-0.89119	88.49	86.71	88.50	86.72
20 - South Hydrant Fire	N/A	38.416	1.6672	89.10	86.44	0.000	5.000	2.66575	2.66575	2.167	1.15039	89.10	86.44	89.12	86.46
20 - North Hydrant Fire	N/A	-33.217	-1.0779	102.58	100.61	0.000	5.000	1.97433	1.97433	2.167	-0.44484	102.58	100.61	102.59	100.62
20 - South Hydrant Fire	N/A	38.385	1.2456	102.89	100.47	0.000	5.000	2.41600	2.41600	2.167	0.57417	102.89	100.47	102.90	100.48
20 - North Hydrant Fire	N/A	-33.186	-0.8364	108.93	106.87	0.000	5.000	2.06171	2.06171	2.167	-0.24325	108.93	106.87	108.94	106.87
20 - South Hydrant Fire	N/A	38.359	0.9667	109.10	106.79	0.000	5.000	2.30321	2.30321	2.167	0.31394	109.10	106.79	109.10	106.80

Figure 3: Output window with multiple scenarios displayed

Library Connections

In the lower left of the Output Control window there is a Library checkbox. If this box is checked, your Output Control parameters are setup as determined by the library to which you are [connected](#). This is referred to as an *active* library. To make it inactive, uncheck the box or change one of the Output Control settings controlled by the library.

If the check box is unchecked, but enabled, you are connected to a library but the settings are not being passed to the Output Control window. The library is thus *inactive*. To make it active, check the box then click the OK button.

If the check box is disabled, there is no connected library.

Command Buttons

There are eight buttons at the bottom of the Output Control window. Fathom has built-in default parameters, units and settings which you can choose by clicking the Fathom Default button. You can also develop your own settings, tailored to your project or industry, and have these used by default (instead of Fathom's defaults). To make your own default, first select the output parameters, units and settings you would like to use then click the Set As Default button. Your settings will be saved and will be used each time any new project is initiated. If you make changes to the settings, and want to get back to your defaults, click the User Default button. The saved settings are updated only when you click Set As Default.

You can save the output settings (except for the selected pipes/junctions and junction deltas) to a file by pressing the Save Control Format button and entering a file name. These settings are loaded again by pressing the Load Control Format button and choosing the file name. For example, you may have a final report format that is always desired. You can load this format before generating final results. If you have another format you use for reviewing model accuracy, you may want a larger number of parameters in the output.

The format files you create can be placed on a network for sharing among a group or company, or incorporated into a company-wide library, allowing standardized reporting.

If you have made changes you don't want to keep, click the Cancel button. Click OK to use the setting you have specified.

Related Topics

Output Window.....	413
Output Parameters.....	424
Applied Standards.....	426
Use the Output Window Sort Feature.....	802
Note: No scenarios have been set in the Multi-Scenario Output display.....	788

Output Parameters

A variety of parameters are available to display in the Output window depending on the junctions and loss models used in the model.

A description of each parameter may be seen by double-clicking the header in the Output Window table, and is also displayed below the Output Parameters table when that parameter is selected in the [Output Control window](#) (Figure 1).

Additional information can be found below for some of the output parameters.

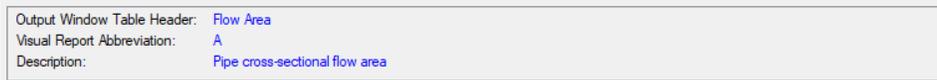


Figure 1: A description is provided at the bottom of Output Control for the selected parameter

Mass Flow Rate Net At Junction

Displays the total mass flow rate exiting the system through the junction, which is useful to display the flow exiting through junctions such as a spray discharge. In other words, the Mass Flow Rate Net At Junction is equal to the flow rate into the junction from the pipes minus the flow rate out of the junction into the pipes.

Mass Flow Rate Through Junction

Displays the total mass flow rate through the junction for junctions connected to only two pipes. If the junction is connected to more than two pipes this parameter will be displayed as N/A. When more than two pipes are connected the user should refer to the output for the pipes connected to the junction in order to view the flow rate through the pipes. The user can add the "Mass Flow Rate Net At Junction" to view flow exiting the system rather than flowing out of the junction into a pipe.

Pressure Loss Static Friction Component

Static pressure drop due to the [Fittings & Losses](#) entered in the Pipe Properties window. Excludes pressure drop due to pipe wall friction.

Pressure Loss Static Friction Total

Static pressure drop due to friction from the wall and the [Fittings & Losses](#) entered in the pipe. This value will be equivalent to the stagnation pressure drop due to friction because the velocity along the pipe is constant.

Pressure Loss Static Friction Wall

Static pressure drop due to friction from the wall. Excludes static pressure drop due to friction from Fittings & Losses in the pipe. This value will be equivalent to the stagnation pressure drop due to wall friction because the velocity along the pipe is constant.

Volumetric Flow Rate Net At Junction

Displays the total volumetric flow rate exiting the system through the junction, which is useful to display the flow exiting through junctions such as a spray discharge. In other words, the Volumetric Flow Rate Net At Junction is equal to the flow rate into the junction from the pipes minus the flow rate out of the junction into the pipes.

Volumetric Flow Rate Through Junction

Displays the total volumetric flow rate through the junction for junctions connected to only two pipes. If the junction is connected to more than two pipes this parameter will be displayed as N/A. When more than two pipes are connected the user should refer to the output for the pipes connected to the junction in order to view the flow rate through the pipes. The user can add the "Volumetric Flow Rate Net At Junction" to view flow exiting the system rather than flowing out of the junction into a pipe.

Related Topics

Output Window.....	413
Output Control.....	418
Applied Standards.....	426
Use the Output Window Sort Feature.....	802

Applied Standards

When an applicable standard is used in a calculation, an additional tab appears in the general area of the [Output Window](#). This tab indicates what standard(s) were used for what calculations in the model.

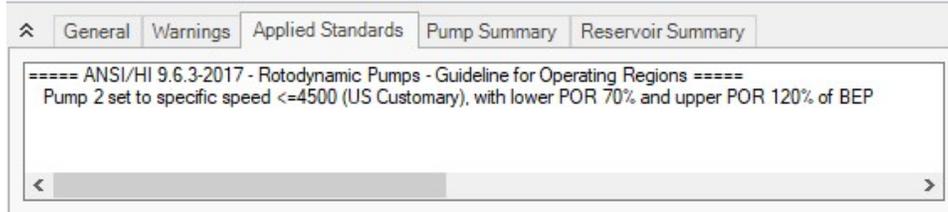


Figure 1: Applied Standards when Pump Viscosity Corrections are used in a model

Related Topics

Output Window.....	413
Output Control.....	418
Output Parameters.....	424
Use the Output Window Sort Feature.....	802

Graph Results Window

The Graph Results window, shown in Figure 1, allows you to generate high quality printed graphs.

The Graphing Panel offers [several types of standard graphs](#) for pump vs. system, profile, transient ([XTS Module Only](#)), and slurry system ([SSL Module Only](#)) information. Users can also control graph content, plotting information for multiple parameters, pipes, paths, or scenarios simultaneously.

Results can be graphed as linear or log versus linear.

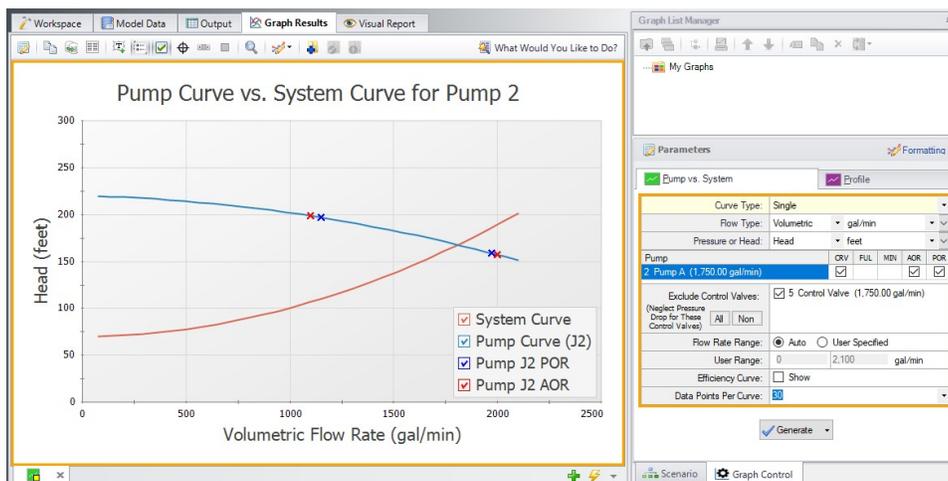


Figure 1: The Graph Results window offers full-featured plot generation

Graph List Manager

Allows the user to create, update, and even replace unique graphs that can be regenerated with new data. Multiple graphs can be displayed at the same time and organized into separate Folders. The default folder is named “My Graphs” and more folders can be created using the “New Folder...” icon on the Graph List Manager Toolbar or by right clicking on “My Graphs” and selecting “New Folder...”

Once a folder is created and selected, a Graph can be added to its list by selecting the “Add Graph to List” icon found on the Graph Results Toolbar. This can also be done by right-clicking anywhere on the graph. Once Graphs are in a Folder individual Graphs can be generated in unique tabs or each Graph in a Folder can be displayed in a single tab by right clicking on the Folder and selecting “Load Current Folder Graphs in New Tabs” or “Load Current Folder Graphs in Same Tabs”. These options are available in the Graph List Manager toolbar as well. Other options that can be found by right clicking on a Folder or Graph in the Graph List Manager or on the toolbar are:

- **Load Graph Item in Current Tab** – Regenerates the graph in the currently active tab.
- **Load Graph Item in New Tab** – Creates a new tab and regenerates the graph there.
- **Load Folder items** - Regenerates all the Graphs in a Folder in separate tabs, in the same tab side by side, or in the same tab as stacked graphs, depending on the selection.
- **Keep Folder Display Layout** - Saves the current layout when multiple graphs are loaded on the same tab.
- **Reset Folder Display Layout** - Resets the Graph List Folder to the default layout when multiple graphs are loaded on the same tab.
- **Close Open Folder Items** - Closes any graphs that were opened from the current folder.
- **Delete Folder Items** - Deletes all graphs in the current folder.
- **Print Preview All Graphs in Folder** - Displays a print preview of all items in the current folder.
- **Move Graph Item Up** – Moves the selected Item up one level.
- **Move Graph Item Down** - Moves the selected Item down one level.
- **Rename Graph Item** - Changes the name of the Item.
- **Duplicate Graph Item** – Creates a copy of the selected Item with a new name.
- **New Folder** – Creates a new Folder where more Graphs can be saved to.
- **Delete Graph Item** – Removes Item from the Graph List Manager.
- **Move Graph Item To Folder** – Moves the selected Item to a previously existing Folder or a new Folder.

Graph Results Toolbar

The Graph Results Toolbar also has many options that are available right clicking on the Graph Results space at any point. These options are:

- **Select Graph Parameters** – Select the type of Graph to create and its parameters.
- **Add Graph To List** – Saves the Graph to the currently active list allowing it to be regenerated in the future.

- **Update Current Graph In List** – Saves any changes to the current graph that have been made. (e.g. Changing the Graph Title or Line Style). Note that it is preferred to use the Replace Graph in List option if changes to the parameters or objects/paths being shown have been made.
- **Replace Graph In List** – Replaces the graph highlighted in the Graph List with the current graph that is shown.
- **Copy Graph Image** – Copies a .jpeg of the Graph to the clipboard
- **Copy X-Y Data** – Copies the X-Y data used by the Graph to the clipboard
- **Show Data** – Displays the X-Y data used to generate the graph in a column to the right of the graph
 - Right-clicking on the x-y data set provides further options to find the maximum or minimum in the data set, create an annotation on the graph for a data point, or to copy an individual data point
- **Create Annotation** – Add an annotation to the graph that can be formatted and saved to be reloaded with the Graph in the future.
- **Show Legend** – Toggles the legend off and on.
- **Show Check Boxes in Legend** – Adds a check box in the legend that allows the user to selectively turn off and on certain curves in a Graph without having to recreate a new Graph.
- **Show Cross Hair** – Displays and cross hair that shows the cursors X-Y values along the X and Y axis. This also displays the nearest data point X-Y values in a tooltip. If this feature is used at the same time as "Show Data" the Cross Hair will focus on that highlighted data point and display its values in a tooltip. The user can also change what data point is highlighted with the arrow keys and the Cross Hair and tooltip will move to that datapoint.
- **Show Range Finder** – Adds a toolbar at the bottom of the graph that displays the entire X axis, but allows the user to zoom in to a specific range by clicking and dragging the handles on either side of the range finder.
- **Show Junction Locations** – Add a vertical line with the number of the junction labeled at the location of the junction along the flow length.
- **Manage Junction Locations** - Allows the user to show/hide the labels for each junction on the profile graph when Show Junction Labels is enabled.
- **Zoom to 100%** - Resets the focus of the graph to 100%
- **Format** – Allows the user to change the Graph's format. Including
 - Header, Axis, and Legend names and fonts.
 - X and Y scaling
 - Line color and thickness

Graph Outline Colors

The active graph will always have an outline which indicates what state it is in.

- **Orange** – Graph not added to list
- **Dark Green** – Graph added to list
- **Light Green** – Graph added to list, but changes have occurred. These can be saved and make the graph dark green by selecting "Update Current Graph in List" on the toolbar.

Graph Tab Icons

Each Graph Type has a specific color associated with it. This color will display in the box on the tab for the Graph. The colors are below:

- **Green** – Pump vs. System
- **Profile** – Purple
- **Transient** – Light Blue
- **Slurry System Curve** – Pink
- **Blank Graph** – Gray

These tabs also display if the Graph in the tab is not saved or has been changed from the original Graph List item by showing a small yellow square over the tab's icon. If the current graph has been saved, there will be no yellow square.

Graph Tab Actions

New Tab – Creates a blank tab in which a new type of Graph can be generate and added to the Graph List Manager

- **Tab Actions**
 - Close Active Tab
 - Close All Tabs
 - Close All But The Active Tab
 - Two Vertical Tab Groups
 - Two Horizontal Tab Groups
 - Reset to Single Tab Group
 - Tabs on.... – Selects the location of tabs (defaults to the bottom of the Graph)
- **Graph Tab Selection** – Selects which Graph tab will be active (shown) based on the Graph Name. The user can also switch tabs by clicking on the desired tab.

Profile Cross-Plotting Features

The graphical data can be exported to a file for later import as a cross plot against results for a different model. For example, let's say you would like to see the effect on the system pipe pressure drop when you change a valve loss factor. You could run the Solver for one value of K and export the results to a file. Then you could rerun the Solver for a new value of K and import the results from the previous case for cross plotting. These features are accessed using the Save Graph Data and Import Graph Data command buttons on the Graph Results File window.

The Graph Results window is always erased when you make changes to model input. Rerun the model to graph the new results.

Multiple paths in the same scenario can also be plotted against each other by adding each path to a group, then choosing the "Plot Multiple Paths Using Groups" option as shown in Figure 3.

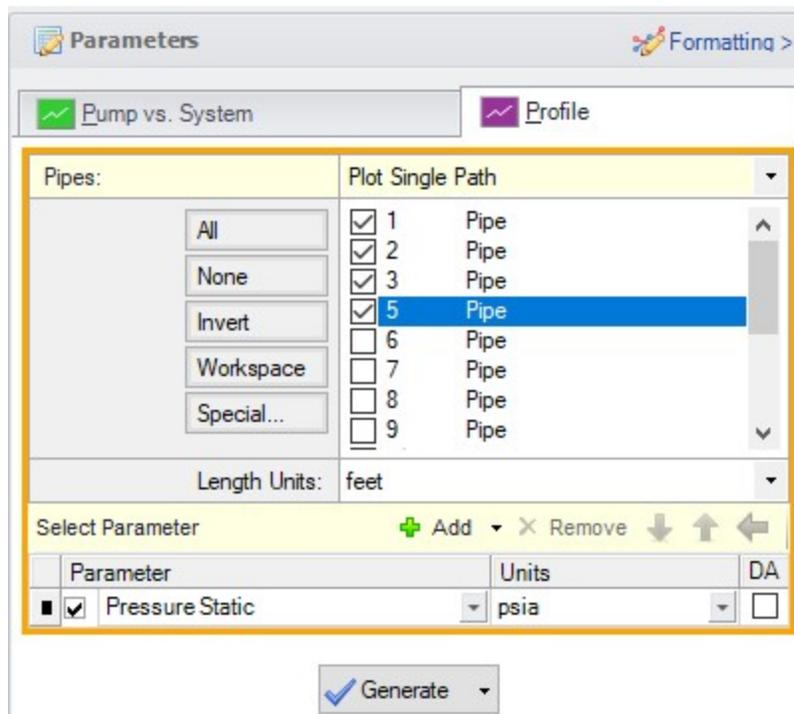


Figure 2: The Select Graph Parameters window controls the Graph Results window

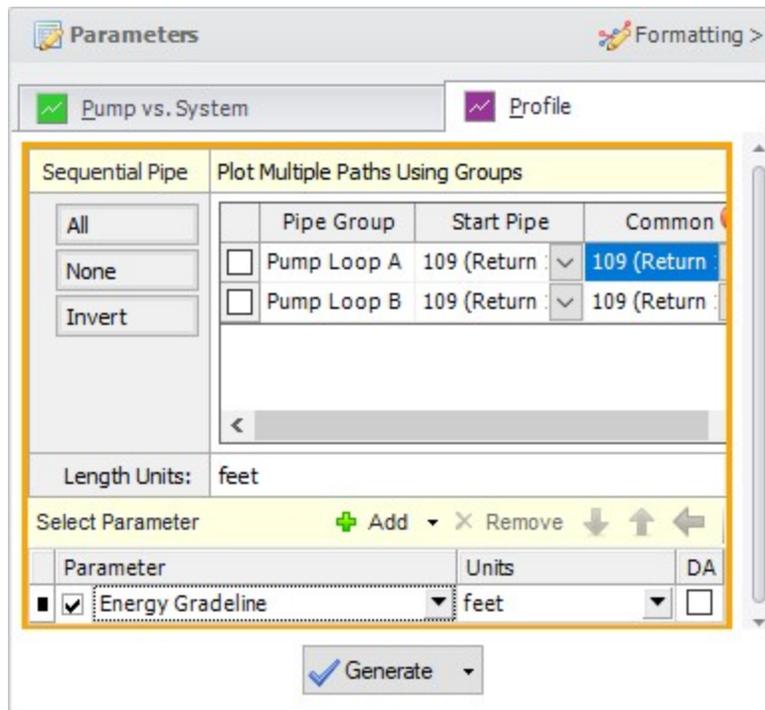


Figure 3: Multiple paths can be cross-plotted using predefined groups

Printing and Exporting

The Graph Results image can be:

- Copied to the clipboard
- Saved to a picture file
- Saved directly to an Adobe PDF file
- Sent to the printer

The x-y data in the Graph Results graph can be:

- Copied to the clipboard
- Exported directly to Excel using the [Excel Export Manager](#)
- Exported to a delimited file suitable for direct importing into spreadsheet software

Related Topics

Interface and User Options.....	359
Menus and Toolbars.....	359
Quick Graph Formatting Window.....	432

Graph Types.....	432
Pump vs. System Curve Methods.....	435
Multi-Scenario Pump vs. System Curves Tutorial.....	440

Related Examples

Beginner - Three Reservoir	823
----------------------------------	-----

Quick Graph Formatting Window

The Quick Graph Formatting window is accessed by clicking the Formatting button in the [Quick Access Panel](#), or from the Tools menu.

The Quick Graph Formatting window allows you to customize the plot by giving you control over the many aspects including:

- Axis titles and font sizes
- Legend text and font size
- Line thicknesses and basic color sets
- Axis min and max values

Changes can be undone by using the Reset button at the bottom of the window, or can be returned to the default formatting by using the Fathom Default or User Default buttons at the bottom of the window.

Related Topics

Graph Results Window.....	426
Graph Types.....	432
Pump vs. System Curve Methods.....	435
Multi-Scenario Pump vs. System Curves Tutorial.....	440

Graph Types

AFT Fathom offers several types of graphs to help users better understand and interpret their data. Each type of graph is given a brief description below.

Pump vs. System Curve

This graphing option allows you to plot a pump curve versus a system curve for a single pump or group of pumps.

Note: The Pump Curve vs. System Curve is only available if there is a pump in the model

You can create pump vs. system curves for complex parallel flow networks. Composite pump curves for pumps in parallel or in series can also be also generated. If you create a composite pump curve, you can

also show single pump curves on the same graph. If efficiency data exists you can cross-plot the efficiency curve.

If POR or AOR have been defined in [Pump Configuration](#), these items can be displayed on the Pump vs. System graph.

If control valves are present in the model the control valves will be excluded from the Pump vs. System curve by default. Excluding the control valves will set them to be fully open for all flow rates. This option is useful to prevent convergence issues.

Pump vs. System curves can be displayed in two ways when parallel pumps are selected. One way is the “traditional” method which is how AFT Fathom displayed them in AFT Fathom 6.0 and before. The second method is called the “enhanced” method and it attempts to determine the flow split on the pumps and adjust the system curve accordingly. It also attempts to create a better composite curve when parallel pumps are dissimilar. For more information see [Pump vs. System Curve Methods](#).

Pump vs. System curves for different scenarios can be plotted together on one graph by adding individual pump vs. system curves to a Multi-Scenario Pump Vs. System Folder, then loading them together in a single Multi-Scenario graph. For detailed information on how to do this see [Multi-Scenario Pump vs. System Curves Tutorial](#).

Profile

This graph category will allow you to plot the selected parameter(s) along a flow path. In order to generate this graph, the specified pipes in the Select Graph Parameters window that comprise the path must form a single, continuous path. If they do not, an error message will appear and inform you which pipes are preventing the path from being continuous. The independent variable in these graphs is the length along the pipe(s).

A path of pipes can be specified in several ways. First, the user can select the pipes in the displayed list. Additionally, you can select the path on the workspace, and then use the Workspace button on the Select Graph Parameters window to automatically select all pipes currently selected on the Workspace. These methods permit that only a single path to be specified.

A profile graph can also be generated directly by selecting a single, continuous flow path in the Workspace, then right-clicking the Workspace and choosing the desired parameter under "Generate Profile Graph."

One or more flow paths can be specified using [groups](#). To do this change the profile plot from "Plot Single Path" to "Plot Multiple Paths Using Groups" at the top of the parameter selection area. Only those groups that consist of a continuous pipe sequence will be displayed as an option in the Select Graph Parameters window.

Multi-Scenario Graphing Features

Starting with Fathom 12, users can plot information from multiple scenarios in the same graph for profile plots (or Transient plots with the [XTS Module](#)). Note [Multi-Scenario Pump vs System curves](#) are generated using a different method.

In the Select Graph Parameters window, the button for 'Multiple Scenarios' allows users to select the scenarios in the model they wish to plot for a particular graph. The same pipe layout must exist for all scenarios being plotted in a graph. Design alerts cannot be included in multi-scenario plots.

The active scenario must be included in a multi-scenario graph. When creating a new graph, users are not able to deselect the current scenario. When a user loads a saved Graph List Item that does not include the active scenario, Fathom will add the active scenario into the Graph List Item, but will not save the active scenario into it.

Scenarios can be selected using the Multi-scenario... button as shown in Figure 1 below.

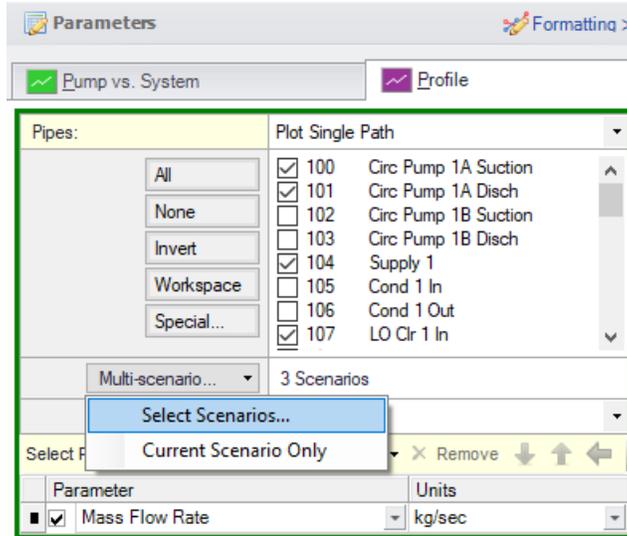


Figure 1: Graph Parameters window showing Multi-scenario selection options

Transient

([XTS Module Only](#)) The Transient allows you to create plots of pipe and junction output with time as the independent parameter. See [Transient Graph Results](#).

The overall maximum and/or minimum values can be plotted. Both of these graph types support [animation](#).

To graph the results at a particular point in time, select the animation option and use the tools provided to display the desired time.

Slurry System Curve

([SSL Module Only](#)) Special system curves can be generated for slurry modeling. See [Graphing Slurry Results](#).

Related Topics

Graph Results Window.....	426
Quick Graph Formatting Window.....	432
Pump vs. System Curve Methods.....	435
Multi-Scenario Pump vs. System Curves Tutorial.....	440

Related Examples

Heat Exchanger System	866
-----------------------------	-----

Pump vs. System Curve Methods

When plotting pump and system curves with two or more parallel pumps, AFT Fathom offers two methods: Enhanced and Traditional.

It should be understood by the engineer that it is not possible to create an unambiguous composite pump and system curve for parallel pump systems. The reasons why are discussed here.

First, let's discuss the head curve. A composite pump curve shows a single head curve with values at particular flow rates. However, when there are parallel pumps with unequal head curves or even equal head curves with unequal flow distribution, it is not clear what flow rate to use in the composite head curve. Further, with unequal flow distribution, different heads for each parallel pump can result. But the composite head curve shows only a single head for any given flow. What head is this? The maximum, minimum, arithmetic average, flow rate weighted average, or some other value? A composite curve has uncertainty over what is meant by the composite flow rate and composite head.

Composite system curves have similar issues to composite head curves. The entire system performance is boiled down to a single head vs. flow rate curve. What flow rate is being referred to and what head is being referred to which goes with a given flow rate?

AFT Fathom attempts to create as meaningful and helpful composite pump and system curves as possible, but the fact remains that the curves AFT Fathom creates are ambiguous because of the nature of composite head vs. system curves.

Enhanced

The "enhanced" method was introduced in AFT Fathom 7.

For composite head curves, it assumes that the parallel pump head curves are added to the right with respect to flow rate. No attempt is made to apportion flow splits among the pump curves. With the head curves added to the right, there is a unique head value for every flow rate. In reality, the parallel pumps may not work this way, depending on the flow split among the pumps and whether or not the head curves are identical and whether there are check valves at each pump discharge. This is, however, a reasonable approximation and represents the historical way that parallel pump head curves are added together.

No matter what method is used for creating a composite system curve, composite system curves are created by fixing some flow rate to each pump and then determining the required head from the pump to deliver the assumed flow rate.

The enhanced system curve takes the flow split at the operating point and assumes that the flow split remains the same at all flow rates. It collects the required head for each pump at each flow rate and then averages these heads together arithmetically.

If the pump head curves are identical and the flow split among pumps is proportional, this method will give the same results as the Traditional method.

Traditional

The “traditional” method was used in AFT Fathom 5.0 and 6.0. The traditional method assumes that the flow splits equally among all pumps at all flow rates, regardless of whether pump curves are identical or not.

The composite head curve is created by taking the composite flow rate, splitting it equally among each pump, obtaining the head for each pump at the split flow rate, and then taking the arithmetic average of the pump heads.

The composite system curve is obtained by taking the composite flow rate, splitting it equally among each pump, running the system model with these flow rates, and then taking the arithmetic average of the required heads for each pump.

Related Topics

Graph Results Window.....	426
Quick Graph Formatting Window.....	432
Graph Types.....	432
Multi-Scenario Pump vs. System Curves Tutorial.....	440

Basic Graph Tutorial

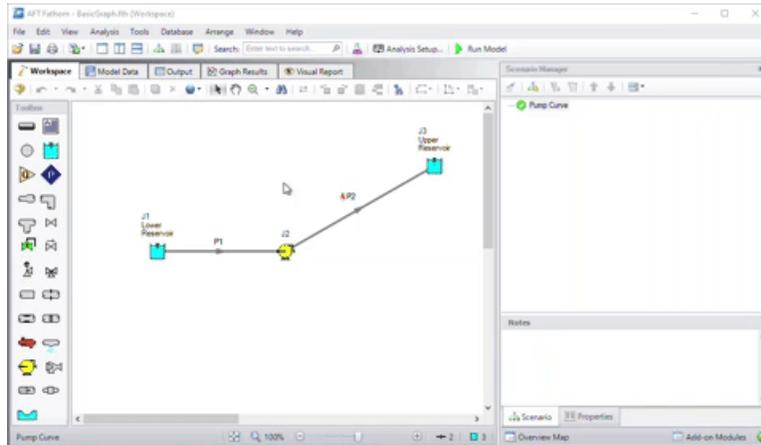
Graphs are powerful analysis tools. AFT Fathom has special tools to create and save Graphs and regenerate them later even if the output data has changed due to model changes. Follow the tutorial below to make and manage graphs from one of the example models.

Create and run a fully defined model. This tutorial uses the Sizing a Pump example model, available in the standard installation of AFT Fathom.

To skip to different sections of this tutorial, use the links below:

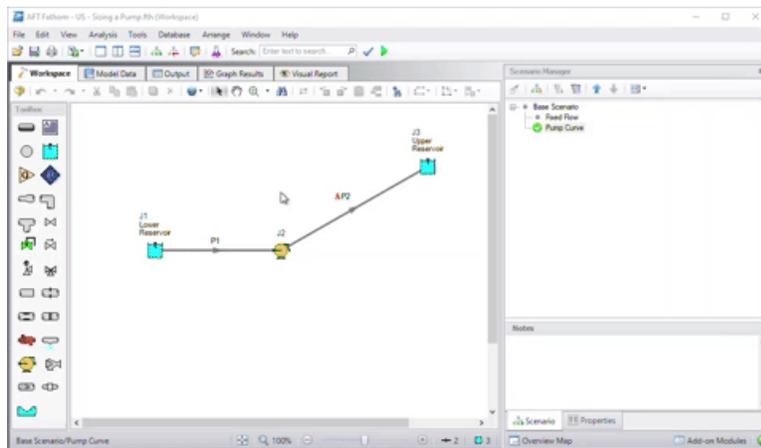
Pump vs. System Curves

Using a model with output, navigate to the Graph Results window. On the right in the Quick Access Panel, select the Pump vs. System tab and select the desired parameters (for the example, the defaults have not been altered) in the Parameters area. After the parameters have been selected, click Generate, and the graph will appear on the left.



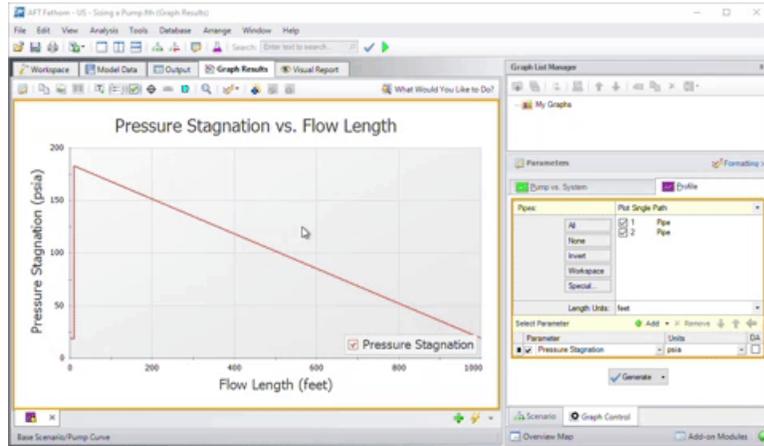
Profile Graphs

Using a model with output, navigate to the Graph Results window. On the right in the Quick Access Panel, click the Profile tab. This graph type displays the selected parameter along the length of the flow path. Select both pipes in the system and change the parameter to Pressure Stagnation. Using the default units of feet for length and psia for pressure, click Generate and the graph will appear on the left.



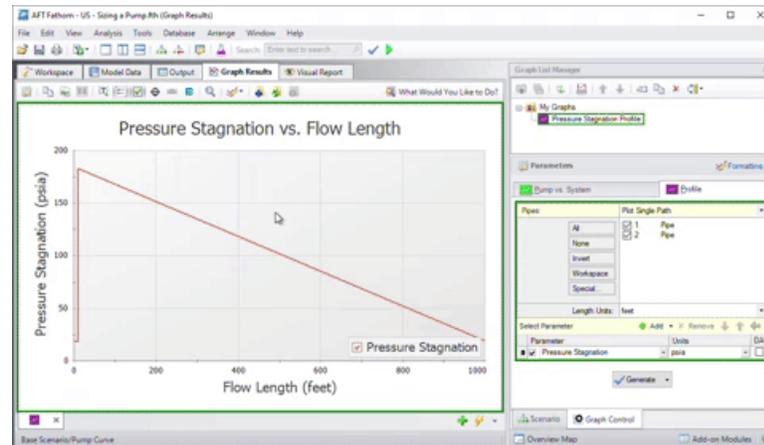
Saving Graphs

After generating a graph, save the graph by selecting the Add Graph To List shortcut on the Graph Results Toolbar. Give the graph a name and it will appear under My Graphs in the Graph List Manager on the right. Easily regenerate this graph with the same parameters and format if the output changes by double-clicking on the graph name. The border of the Graph has also changed to dark green along with borders of the parameters and the name of the Graph in the Graph List Manager. This indicates that that Graph is current, saved, and that the displayed parameters are the ones used to generate the Graph.

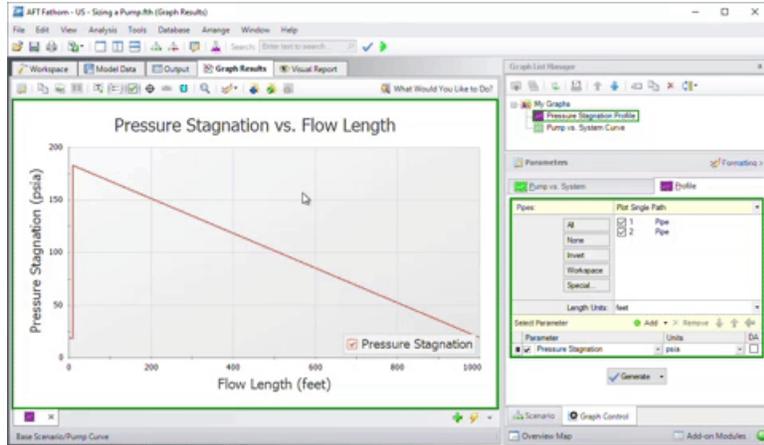


Creating Multiple Graphs

Create multiple graphs stored on their own tabs by first clicking on the green plus button on the bottom right of the graph window to create a new graph tab. This creates a new graph tab. Generate a graph and then toggle between the graphs by selecting the tabs on the bottom of the graph window. Add as many graphs as desired.

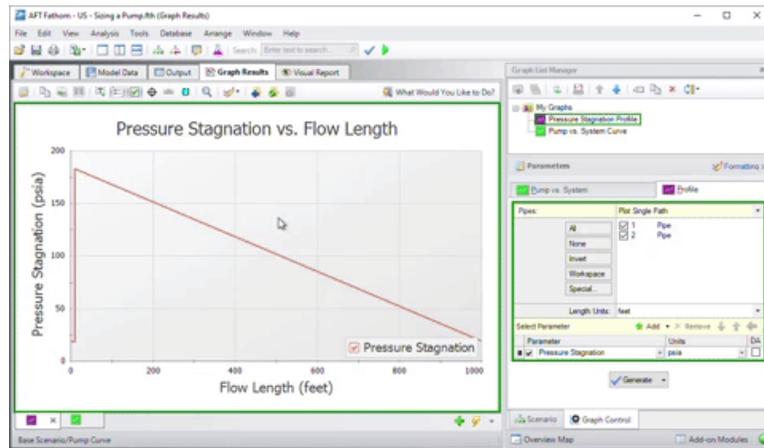


Alternatively, double-click a saved graph in the Graph List Manager in the Quick Access Panel to automatically generate that graph on a new tab.

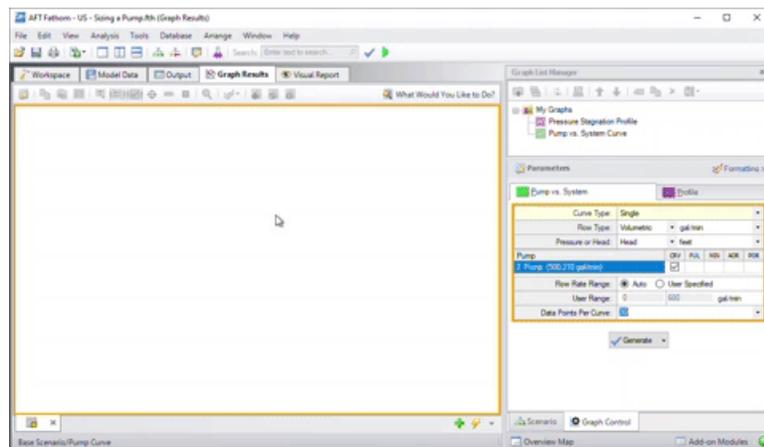


Displaying Multiple Graphs Simultaneously

To view multiple graphs in the same window, right click on a graph tab and select either Two Vertical Tab Groups or Two Horizontal Tab Groups. The graphs will then display next to each other such that both graphs can be viewed simultaneously.

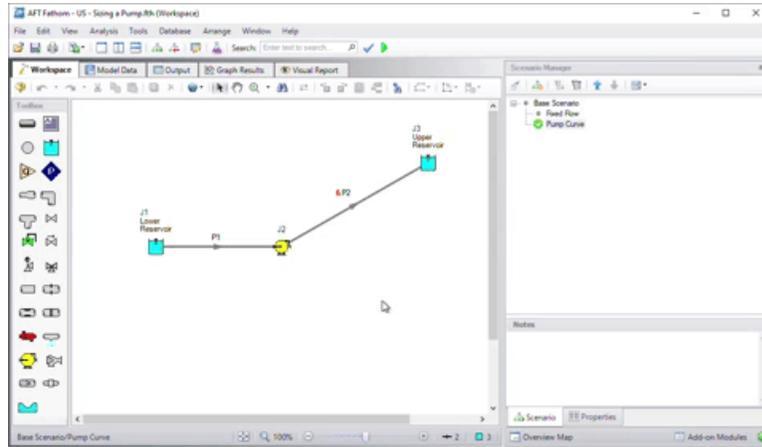


Alternatively, right-click on the My Graphs folder in the Graph List Manager, choose Load Folder Items, then click Same Tab. This will generate all graphs in the My Graphs folder into the *same tab* (as opposed to displaying *multiple tabs* simultaneously as described previously).



Creating Graphs from the Workspace

Some graphs can also be generated directly from the Workspace after running the model. Generate the Pressure Stagnation vs. Flow Length graph from the Workspace by selecting the flow path on the Workspace (note that you can select multiple pipes by holding the SHIFT button on the keyboard while selecting each pipe). Right click on the Workspace, choose the Generate Profile Graph for Pipes option, then click Pressure Stagnation.



Multi-Scenario Pump vs. System Curves Tutorial

When a model file has multiple scenarios that have pumps in them, a graph containing multiple pump curves and/or system curves can be generated for the purpose of comparison. To create a multi-scenario pump vs. system curve graph, two options are available:

- **Standard Method:** Save pump vs system curve graphs for each individual scenario to be included in the graph. This allows the user to define settings such as the flow range for the pump vs system curve on an individual scenario basis, but is more time consuming.
- **Batch Run Method:** Create a template pump vs system curve graph, then set up a batch run to automatically generate the pump vs system curves for all scenarios to be included in the graph. This allows for a faster process.

Standard Method

1. In the Graph List Manager select "New Folder". Provide a name for the folder, and check the box to "Make Multi-Scenario Pump Vs. System Folder", as is shown in Figure 1. This folder will be used to save the curves from the individual scenarios that will be included in the multi-scenario graph.

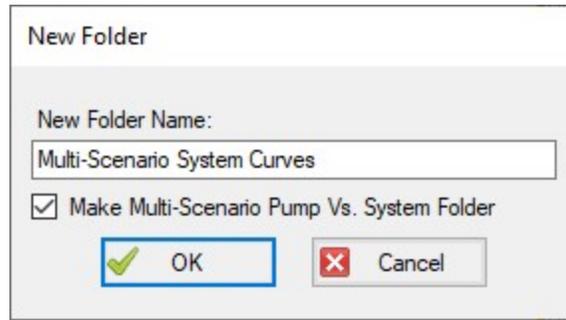


Figure 1: Creating a Multi-Scenario Pump Vs. System Folder

2. Load the first scenario you want to add to the graph, and run the scenario so that Output is present. Generate a Pump vs. System Curve for the scenario.
3. Add the Pump vs. System Curve to the Multi-Scenario Pump Vs. System Folder. This can be done by selecting the Multi-Scenario folder in the Graph List Manager, then clicking the Add Graph to List button, and naming the graph. The graph should now be shown subordinate to the folder, with the scenario name listed next to it in red, as is shown below.

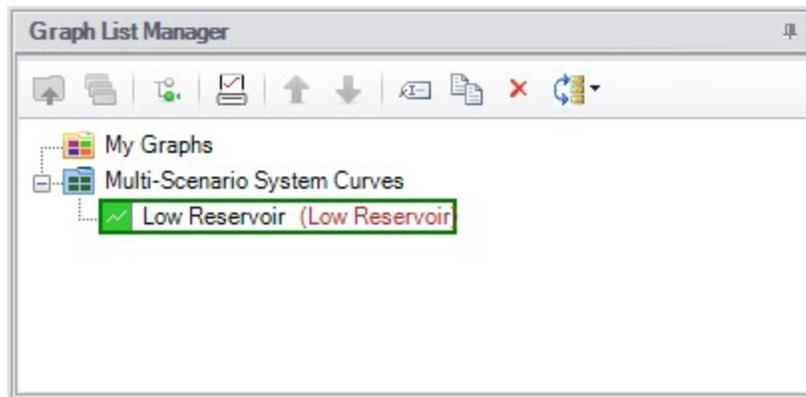


Figure 2: The name of the scenario that each Pump vs. System Curve was created in is displayed in the Graph List Manager

4. Repeat steps 2 and 3 for all scenarios that will be included in the graph.
5. When all of the individual Pump vs. System curves have been added to the folder, right-click on the folder name and choose "Load Multi-Scenario Graph," as is shown in Figure 3.

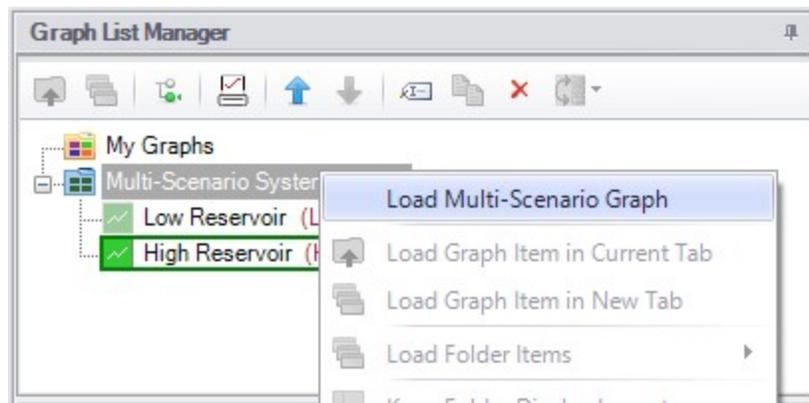


Figure 3: A Multi-Scenario Graph can be loaded from a Multi-Scenario graph folder

6. In the Multiple Scenario Graph Selection window select which curves you would like to include in the graph, then choose Generate Graph as is shown in Figure 4.

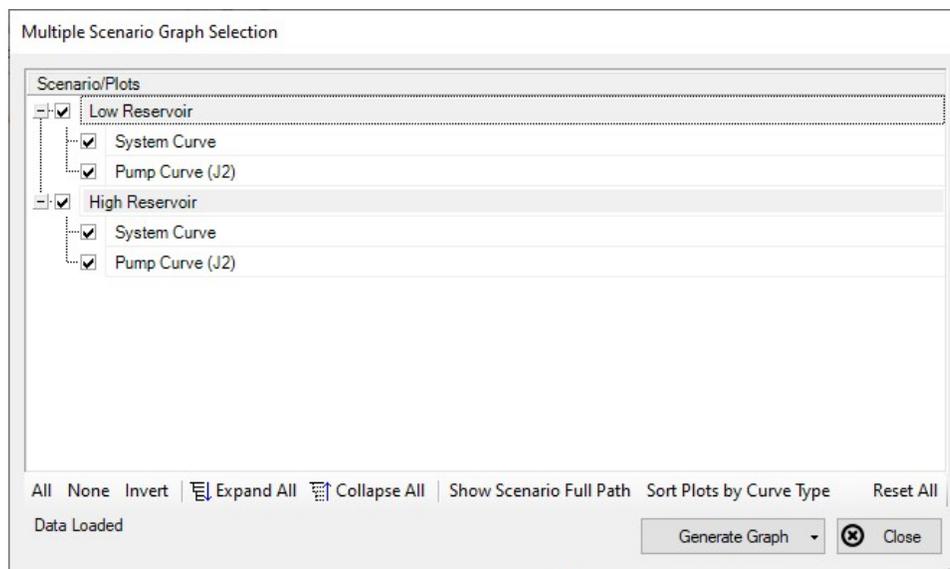


Figure 4: Multiple Scenario Graph Selection

7. The selected curves will now be displayed on a single graph, as is shown in Figure 5.

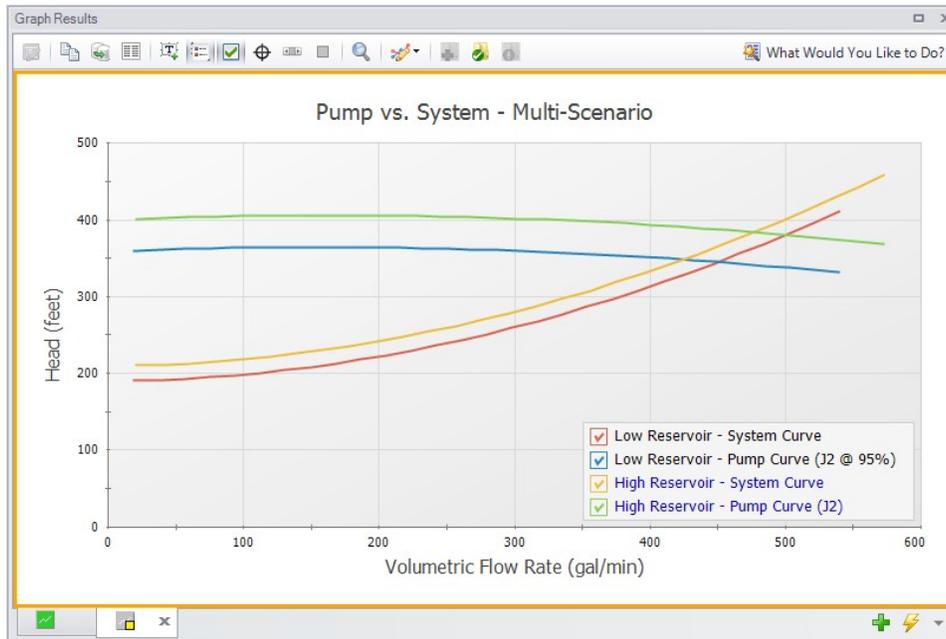


Figure 5: Multi-Scenario Pump vs. System curve

Batch Run Method

To speed up the process outlined above when making Multi-Scenario Pump vs. System curves for more than a handful of scenarios, the [Batch Run](#) tool can be used. Follow the steps below to create the curves.

1. In one of the scenarios being included in the Multi-Scenario Pump vs. System curve, generate a single scenario Pump vs. System curve and save it as a standard Graph List item. This curve will be used as a template for all the other scenarios, and should be for a pump common to all scenarios.
2. Launch the Batch Run window found in the File menu. Select the option to have the Batch Runs create Multi-Scenario Pump vs. System curves, and select the template Pump vs. System curve as shown in Figure 6. If you have multiple templates, all options will be shown for selection.

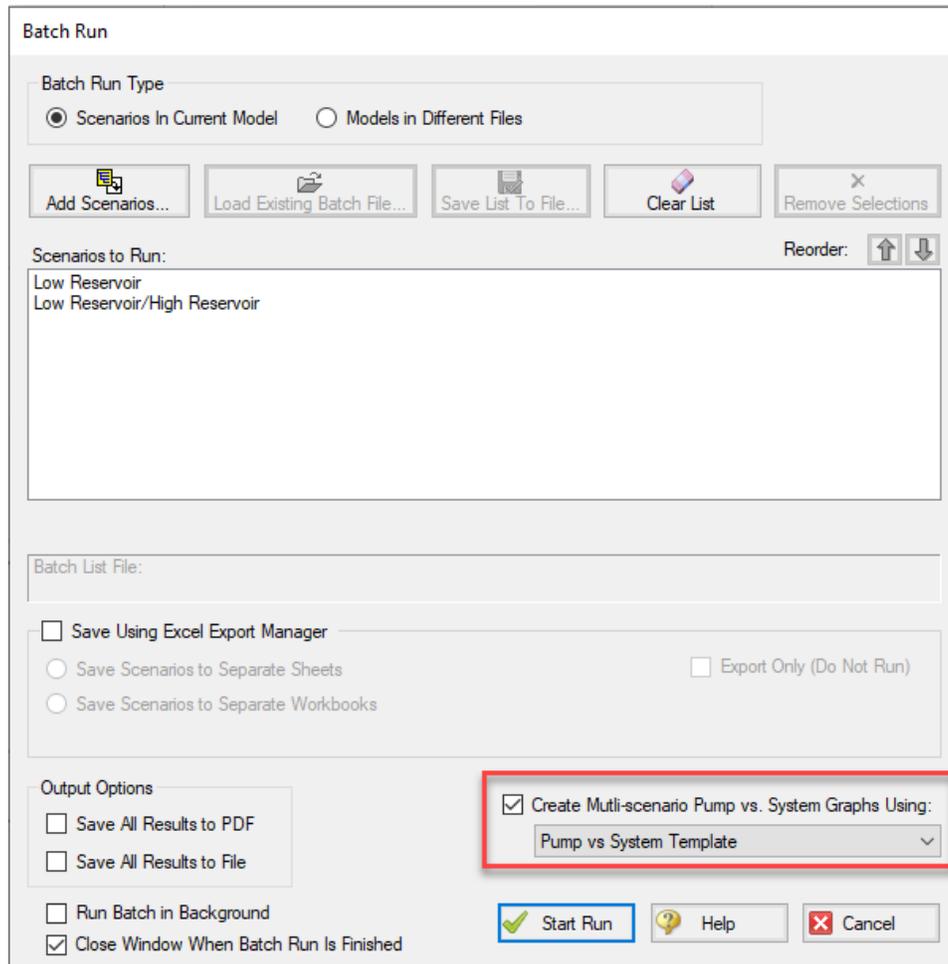


Figure 6: Multi-scenario batch run option enabled in the Batch Run window

3. Add the desired scenarios to run to the list and start the run.
4. Once the run is complete, navigate to the Graph Results tab. In the Graph List Manager, there will be a Multi-Scenario Pump System Curve folder named with the date and time of the run. Right-click on the folder name and choose "Load Multi-Scenario Graph," as is shown in Figure 7.

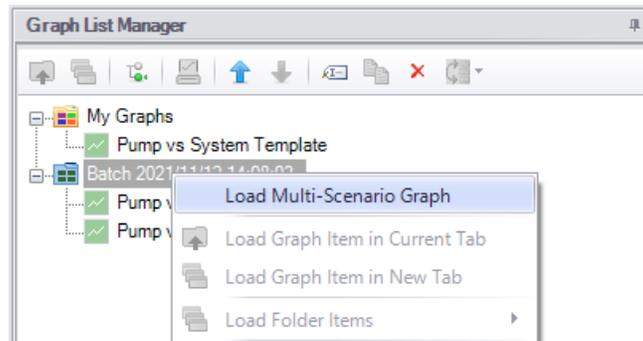


Figure 7: A Multi-Scenario Graph can be loaded from a Multi-Scenario graph folder

5. In the Multiple Scenario Graph Selection window select which curves you would like to include in the graph, then choose Generate Graph as is shown in Figure 8.

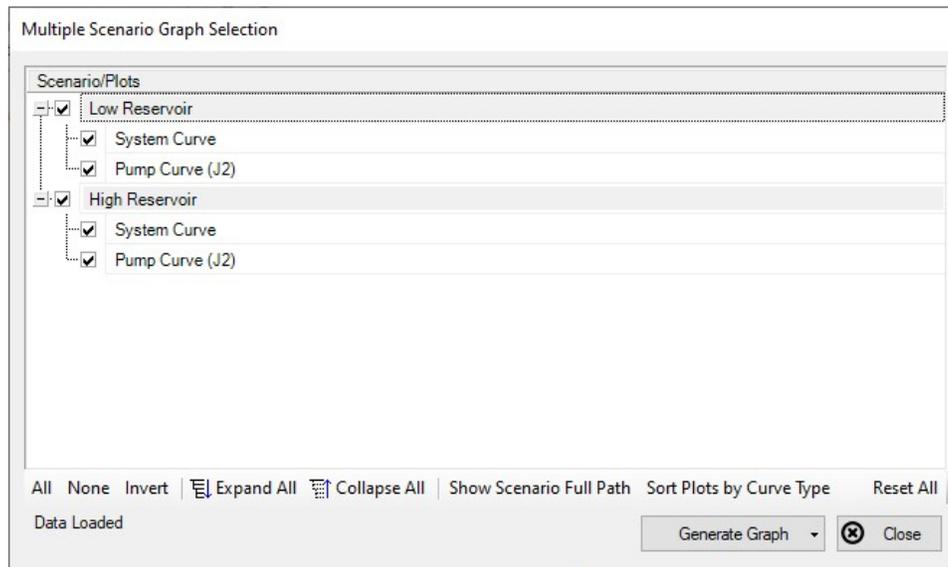


Figure 8: Multiple Scenario Graph Selection

6. The selected curves will now be displayed on a single graph, as is shown in Figure 9.

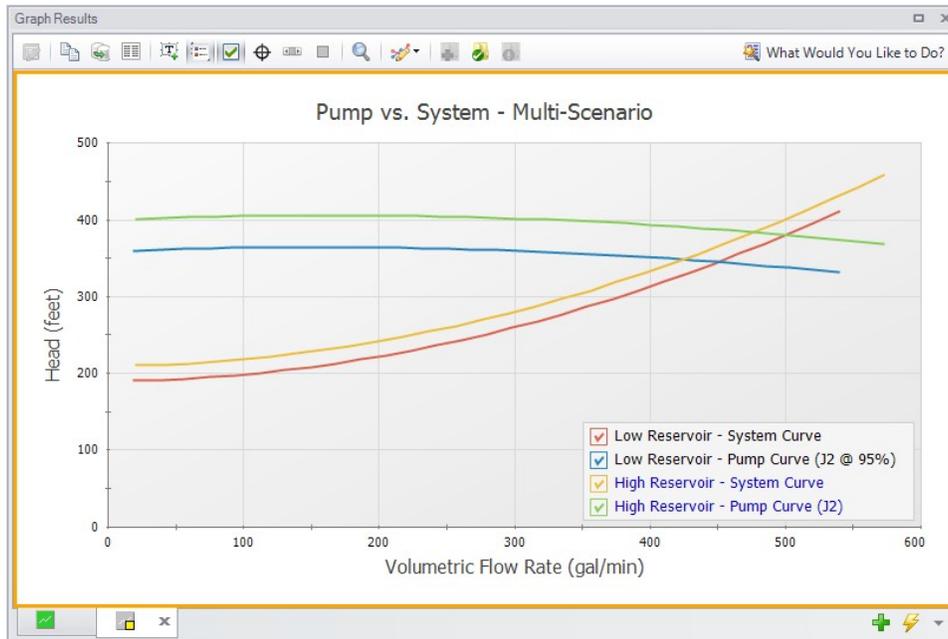


Figure 9: Multi-Scenario Pump vs. System curve

Related Topics

Graph Results Window.....	426
Quick Graph Formatting Window.....	432
Graph Types.....	432
Pump vs. System Curve Methods.....	435

User Options Overview

An array of User Options are provided in a single window that can be found in the [Tools](#) menu.

The General section of the User Options window provides the following options:

- Choose to show the Startup Options window at startup
- Choose to show the New Version Welcome window at startup
- Enable orthogonal drawing tips
- Enable Confirm Before Clearing Results
- Enable highlighting for required fields in the Properties windows, and choose the highlight color
- Options for opening models with scenarios
- Set a visual theme for the interface
- Select a location for the Scenario Manager
- Choose the number of data points to show in input tables
- Setting parameters for "button flashing" for curve fit buttons on junction Properties windows

User Option Defaults

There are several buttons at the bottom of the User Options window. There are built-in AFT Default parameters, units, and settings which you can choose by clicking the Reset to Fathom Defaults button. You can also develop your own settings, tailored to your project or industry, and have these used by default. To make your own default, first select the parameters, units, and settings you would like to use, then click the Save All As New User Defaults button. Your settings will be saved and will be used each time any new project is initiated. If you make changes to the settings, and want to get back to your defaults, click the Load All User Defaults button. The default settings are updated only when you click Set All As New User Defaults.

If you have made changes which you don't want to keep, click the Cancel button. Click Save and Close to use the settings you have defined.

Related Topics

Interface and User Options.....	359
Unit System.....	447
Workspace.....	449
Parameters.....	452
Files.....	454
Sizing Preferences.....	456

Related Examples

Beginner - Three Reservoir	823
----------------------------------	-----

Unit System

You can set the units you prefer to use in AFT Fathom in the Unit System section. Once selected as the preferred units, these units appear as the default units in all instances in which empty fields exist in the model input. Any model input existing prior to changing the Preferred Units will maintain the units used when the data was input, regardless of changes made to unit preferences. After saving your preferred units, a window appears asking if you would like the output parameters to be displayed in your preferred units. Choose Yes if you do want the output parameters to be displayed in the preferred units, or No if you want the output parameters to be displayed in the units used prior to making changes to the preferred units.

Preferred Units

To customize the units used by AFT Fathom, first select Preferred Units under the Unit System section in the User Options window under the Tools menu.

AFT Fathom allows you to specify certain units from a large selection to be displayed in the main interface.

The primary selection is the Unit System:

- **US** - Only US units will be available
- **Metric** - Only Metric units will be available
- **Both with US Defaults** - All Units will be available, but US units will be chosen by default
- **Both with Metric Defaults** - All Units will be available, but Metric units will be chosen by default

Additionally, a filter can be applied to limit the selection:

- **All Units** - all units for the Unit System will be displayed
- **Common Only** - only "common" units will be displayed
- **Common Plus Selected Industries** - Units particular to certain industries can be included with the common units

If the selection has been limited to Common Only, there may units common for certain industries no longer present. The Additional Units for Industry section allows the user to append additional sets of units to the Common unit set. For example, barrels/day is a common unit in the petroleum industry but not many others. Checking the Petroleum box adds these units to the main interface.

Preferred Units allows fine grain control over what unit in the above selection is displayed as the default when defining model input. For example, it may be desired to have a particular parameter such as pipe diameter default to inches, while keeping other parameters metric. This area allows the user to define such defaults.

Show/Hide Unit Items

For additional control of the available units, select Show/Hide Unit Items under the Unit System section. From here, you can select which units for each Unit Family Type you would like to be available for the model input and output by checking the box next to each unit you would like to be used. Removing units that you do not wish to use as input or view as output can simplify your modeling and reduce errors associated with incorrect units.

Edit Custom Unit Names

A custom unit name can be assigned to display different names in the interface for units. For example "feet/min²" could be changed to display as "ft/min²".

To assign a custom unit name choose the appropriate Unit Type from the drop-down and enter the Custom Unit Name in the table.

To speed up the process of entering custom unit names, the units can be exported to *.csv and re-imported as follows:

1. Click "Save Units to CSV" and save the file where it will be easy to locate
2. Open the *.csv file in a spreadsheet editing software like Excel, and fill in the desired custom unit names in the Custom Name column

3. Save the *.csv file
4. In the User Options window click "Load Units from CSV" and select the edited *.csv file

Note: To use units with superscripts, make sure the *.csv file is saved with UTF-8 encoding. This can be done in Excel by choosing Save As and selecting "CSV UTF-8" as the file type.

Note: Regions outside the United States that use commas as the decimal separator may have *.csv files that are delimited with semicolons (this was a change made by Microsoft). AFT software requires true comma delimited files. A workaround for such regional users in Excel is listed below.

Workaround for regional users with alternate decimal separators (e.g. Europe):

1. In Excel go to File > Options > Advanced
2. Under the Editing Options heading, uncheck the box for "Use system separators"
3. Change the "Decimal separator" to a period symbol (.)
4. Change the "Thousands separator" to a comma (,)
5. Save the *.csv file
6. Re-check the "Use system separators" option to revert to the default Excel settings

Related Topics

User Options Overview.....	446
Workspace.....	449
Parameters.....	452
Files.....	454
Sizing Preferences.....	456

Workspace

In the Workspace group of the User Options window, you can change the symbols used, default text content, and location of the text on the [Workspace](#). Note that all display changes are made only for the user's convenience/preference and do not affect the model calculations. Additionally, many display options available in the User Options window are also available in individual pipes and junctions.

Grid

A traditional 2-D grid or an Isometric grid can be displayed on the workspace for modeling convenience based on the [Pipe Drawing Mode](#) that is selected. The number of lines in the grid can be adjusted to create a grid that is fine, medium, or coarse. The grid can be hidden or shown by checking or unchecking the Show Grid box accordingly. If the grid is shown, the grid line pattern can be solid or dashed. The pipes and junctions can also be snapped to the grid, which moves junctions and pipe end points to the closest intersection of grid lines.

The Pipe Drawing Mode and the toggle to Show Grid can also be found on the [Arrange menu](#).

Pipes

The display of pipe lines, closed pipes, pipe endpoints, and pipe direction arrows can be adjusted according to your preferences here.

The default pipe line thickness for new layers can be adjusted by selecting the desired pixel value in the box next to New Layer Default Pipe Line Thickness (pixels).

You can close a pipe using the [Special Conditions](#) feature which closes the pipe to flow at both ends. From the User Options window, you can select how you would like a closed pipe to be displayed on the Workspace. The options include a solid line, dashed line, a dotted line, and dashed-dotted combinations. The color of the line is still determined by the visible [Workspace Layers](#).

When building a model, Fathom automatically adjusts pipe endpoints to the centers of junctions for user convenience. If you would like to disable this feature, uncheck the box next to Automatically Adjust Pipe Endpoints to Junction Centers.

Under the Pipe Direction Arrows section you can select the size of arrow you would like to be displayed as a percentage of the pipe thickness. Additionally, you can opt to not show a pipe direction arrow by checking the option Do Not Show Arrow.

Note: The pipe direction arrow is used as a [reference positive flow direction](#) and does not change direction on the Workspace if the calculated flow is in the opposite direction indicated by the arrow. Instead, the flow is reported as a negative flow in the Output.

Junctions

The junction icon styles include classic icons and transparent icon. Classic icons are the icons used in earlier versions of Fathom ending at Version 7 in which each junction is surrounded by a box, and transparent icons are updated icons not surrounded by boxes.

You can also select the icon size and label location for junctions. To adjust the size, click on the box below Icon Size in the Junctions panel and choose the percentage of the default size that you would like the junction icons to be sized. For the junction label location, you can select from any of the eight locations shown by clicking on the selection circle that corresponds to the desired location.

Using the default transparent icon style, the appearance and color of icons for new junctions can be set in the Toolbox, and icon size and color for existing junctions can be set in [Layer Settings](#). Junction icon and rotation can be edited for existing junctions by right clicking the junction on the Workspace and selecting Customize Icon.

Note: Junction and pipe labels can be moved individually on the Workspace by using the mouse to drag and drop the label according to your preferences.

The final option is to enable or disable a dotted outline around a closed junction. If this option is enabled, a dotted box will appear around the junction when it is closed with Special Conditions. If this option is disabled, no dotted box will appear.

Labels

When a pipe or junction is inserted onto the Workspace, the next whole number available at the increment specified in this panel is automatically assigned to the new pipe and/or junction. The default is to increment pipe and junction numbers by one. Therefore, the first pipe you place on the Workspace will automatically be labeled as Pipe 1 (P1 for brevity), the second will automatically be labeled P2, etc. Junctions are labeled in a similar fashion with a "J" in front of the number to indicate it is a junction. However, if you choose a different increment in the User Options window for pipes and/or junctions, the automatic numbering will increment pipes and/or junctions according to the specified number. You can renumber any pipe or junction individually as long as the number is not already used in any scenario of the model. If a number at the increment is already in use, Fathom will skip to the next available whole number. Pipe and Junction label numbers are limited to integers between 0 and 99,999.

Note: A unique label must exist for each pipe and junction on the Workspace. Therefore, there cannot be more than one P1 in the model, but a P1 and J1 can exist simultaneously because the leading letter is different. Pipe and junction numbers are defined globally and not per layer or per scenario.

Whether pipe/junction names and/or numbers are displayed is controlled ultimately by [Workspace Layers](#), but the Pipe/Junction Default Display setting allows the user to set their preferred default behavior.

Users can also choose whether the pipe and junction labels should be moveable at all times, or if they should only be moveable when the user is holding down the shift key with the Allowable Workspace Label Movements setting. Restricting the movement of labels can be especially helpful to users with complicated or dense models, as it is possible to inadvertently grab a label when attempting to reposition or select an icon on the Workspace.

Colors and Fonts

The Colors and Fonts item located within the Workspace group in the User Options window allows you to change the colors of the following parameters seen on the Workspace:

- Pipe and Junction Symbols
- Junction Labels
- Pipe Labels
- New Layer Default Pipe Color
- Selected Items
- Undefined Numbers
- Workspace Background Base Scenario
- Workspace Background Children
- Workspace Background Grandchildren
- Workspace Background 3rd Generation
- Workspace Background 4th Generation +
- Grid

The currently selected color for each parameter is shown to the right of the parameter. To change the color for any parameter, either select the box containing the parameter's color and then click on the Set Color box under the list, double click on the box containing the color you would like to change, or click the "..." button next to the color of the parameter you would like to change. A color window pops up that allows you to choose from 48 basic colors, or to create a custom color by clicking anywhere in the color box and adjusting the slider at the right of the box to create the color of your choice. If you would like to save the custom color for use in other areas of the model, click on any empty box underneath Custom colors: and then click on the Add to Custom Colors box below the color box. When finished with selecting a color, click OK on the bottom left of the Color window.

To have the background color of the children and grandchildren scenarios maintain the same color as the base scenario, click on the Set All Background Colors to Base Color button under the Parameter/Color list.

Other

Workspace Symbols representing pipe or junction special conditions and inflow or outflow are listed in the Other item within the Workspace group in the User Options window. These symbols can be disabled by unchecking the box on the left of the list. Users can also change what symbol will be used by clicking in the Symbol box on the right and typing the new desired symbol.

If there is a background image in a model, the image size can be controlled from this panel. To scale a background picture, enter a Scale Multiplier in the box underneath Background Picture Scaling.

Related Topics

User Options Overview.....	446
Unit System.....	447
Parameters.....	452
Files.....	454
Sizing Preferences.....	456

Parameters

The Parameters section of the User Options window allows you to specify some default parameters and model input when a new pipe or junction is added to the Workspace. Changing any of the parameters in this window adjusts the specified default pipe and/or junction input parameters and/or values to simplify the modeling process and to prevent the user from having to enter repetitive data.

Pipes and Junctions

Pipe and Junctions, located within the Parameters section of the User Options window, allows you to specify default pipe and junction input information.

Under the Default Pipe Data box in this window, you can enable or disable the option Use Most Recent Pipe Size Data. If you enable Use Most Recent Pipe Size Data, the pipe property windows of new pipes added to the Workspace will automatically contain pipe diameter, material, and friction information from

the most recently entered pipe. This prevents you from having to re-enter the pipe diameter, material, and friction model information in the case that several pipes require this same input. Disabling this feature leaves the pipe diameter, material, and friction model for each new pipe added to the Workspace as the default specified in the User Options window.

The Default Junction Elevation box allows you to select the default junction elevation you would like for all new junctions added to the Workspace. If you select Unspecified in this box, the junction elevation field in the [Junction Properties window](#) remains empty until you specify each junction elevation individually. If you select Use Most Recent, the junction elevation of a new junction added to the Workspace is shown as the elevation of the most recently specified junction. If you select Specified, you are able to input an elevation of your choice that will automatically be input into each new junction that is added to the Workspace.

The Default Reservoir Pipe Specification box allows you to specify the default option selected for how the height at which a pipe connected to a reservoir is specified. Selecting Pipe Elevation requires you to input the pipe elevation with respect to the reference elevation. Therefore, if a pipe is located 100 feet (30.5 m) from the reference elevation, or elevation= 0 ft (0 m), then you would input 100 ft as the pipe elevation when Pipe Elevation is selected in the [Reservoir Properties window](#). Selecting Pipe Depth requires you to input the pipe elevation with respect to the level of fluid in the tank. Here, if a pipe is located 10 feet from the top of the fluid level in the tank, you would input 10 ft. Note that the same results will be calculated regardless of whether you specify Pipe Elevation or Pipe Depth as long as the height of the pipe is defined at the same height with respect to the system in both cases.

The Default Control Valve Failure option allows you to choose the default behavior if a control valve is calculated to have failed. This option can be changed on an individual basis in the Control Valve Properties window. See [Control Valve Setpoint Not Achievable](#) for more information.

The [Default Design Factors](#) box allows you to add a default Design Factor to each junction shown in the list on the Pipes and Junctions window. Setting a default design factor for junctions can be especially useful when applying safety factors on given equipment.

Graphs from Workspace

Various parameters can be graphed quickly by right-clicking on a continuous selection of pipes from the Workspace and choosing the Generate Profile Graph for Pipes option. The graph parameters that are available from the right-click menu can be customized.

Parameter Type

Parameter Types, located within the Parameters section of the User Options window, allows you to specify the flow rate type, head/pressure type and Efficiency/Power types you would like as the default option wherever one of those parameter types is defined in the model. The default flow rate type can be set to either mass flow or volumetric flow, the default head/pressure type can be set to either head or pressure, and either efficiency or power can be selected as the default for efficiency/power. For example, when defining a resistance curve to describe the hydraulic pressure loss behavior through a general component, you are required to enter data that describes pressure loss at given flow rates. When defining the pressure loss behavior here, you can either define the flow rate as a mass or volumetric flow rate, and you can define the head/pressure loss as either a head or pressure loss. Adjusting the default option of these parameter types to fit your available data not only helps to make data input more efficient by

preventing you from having to constantly change parameter type, but it also helps to minimize input errors by adjusting the required input to the form of your available data.

Related Topics

User Options Overview.....	446
Unit System.....	447
Workspace.....	449
Files.....	454
Sizing Preferences.....	456

Files

The Files section of the User Options window allows you to specify file locations for saving models, library folders, user settings, and model backup files. You can also customize the model backup file settings here.

File Locations

Under File Locations in the Files section of the User Options window, you have the option to select the default folder in which to save your model. If a default model folder is specified, this location will be the default location to save the model. To specify the default location, click Add below the Default Model Folder box and browse to the location you want to set as the default folder. Once you have selected the folder, click OK. The location of the default model folder will then show in the Default Model Folder box. You can add more than one default folder, but only the folder selected in blue will be the active default folder.

To specify locations for Fathom to search for [library](#) files, click Add under the Library Search Locations box. Browse to the file location that contains the library, select the location, and click OK. You can add multiple folders to be searched. To add additional folders, repeat the process for each folder.

The User File consists of all saved settings and preferences. In order to backup these user settings in a *.ini file, click Add under User Backup Locations. Browse to the file location in which you would like to save these settings. Once you have selected the desired file location, click OK. Note that the User File can be saved in multiple locations. To add additional locations, repeat the process for each folder that you would like to save the settings.

Backup Files & Auto Save

A Backup File is a copy of a previously saved version of your model file (*.fth) that can function as a complete, standalone model. These files are used as a safety measure to provide you with one or more previous versions of the model in the case that the current model is deleted or damaged.

The Number of Backup Files (i.e. the number of previously saved versions of the model) and the frequency that the model is saved can be adjusted. You can opt to save no Backup Files by choosing "0" in the Number of Backup Files Created dropdown.

After the model has been saved initially, subsequent instances of saving the model will create a new Backup File until the specified maximum has been reached. The most recent Backup File will always contain the file extension *.bakF-001. Larger numbers appended to the file name indicate older backups:

- Model File.fth - Current and most recent version of the model file
- Model File.bakF-001 - Latest backup (the second most recent version of the model file)
- Model File.bakF-002 - Older backup
- Model File.bakF-003 - Oldest backup

If you need to revert to a Backup File and convert it back into a standalone model, add the extension .fth to the Backup File (the file name should now appear as Model File.bakF-001.fth), and open the file in Fathom. From there you can re-save the file or overwrite your existing model file.

The model file can be saved periodically by enabling the checkbox for Automatically Save Model and specifying a frequency in the dropdown box.

Create Recovery File

A Recovery File (*.Recv-FTH) is similar to a Backup File but is saved periodically at the specified interval. It allows the user to recover unsaved changes in the model file prior to the application crashing or closing abnormally. The Recovery File is written to the same location as the model file, and each time the specified interval passes it will be overwritten to contain the most recent state of the model. If the application is closed normally the Recovery File will be deleted. A Recovery File can only be generated for existing model files that are opened.

If a Recovery File exists, the next time the model file is opened a prompt will be shown asking if you want to use the Recovery File:

"A recovery file for this model was found which may indicate that the application closed unexpectedly. Would you like to open this recovery file?"

- **Yes** - This will copy the previously saved model file to a new file with "(Original)" appended to the front, and the Recovery File containing the latest unsaved changes will be opened.
- **No** - The Recovery File will be deleted, and the previously saved model file will be opened. This will open the model file in its last saved state, which may not include the latest unsaved changes if any are stored in the Recovery File.

Encoding

The File Encoding option can be set to "Unicode" in order to use characters outside of the standard English keyboard such as diacritics, cyrillic, etc for text fields such as the pipe/junction name, notes, etc.

Related Topics

User Options Overview.....	446
Unit System.....	447

Workspace.....	449
Parameters.....	452
Sizing Preferences.....	456

Sizing Preferences

The Sizing Preferences section provides options to customize the colors used in the Sizing window, set the location of the Sizing Navigation Panel, and enable options for creating continuous/discrete candidate sets.

Note: This section of User Options will only be shown if the [ANS module](#) is activated in Fathom.

Color Options

The "Sizing Option" parameter colors affect the background colors used for the corresponding columns in the "Options to Minimize Monetary Cost" table on the [Sizing Objective panel](#) and the "Sizing and Cost Options" table on the [Size/Cost Assignments panel](#).

The assigned background colors will also change the background of the Sizing Level status button in the [Sizing Navigation Panel](#).

The "Common Size Group" parameter colors affect the background colors used in the "Pipe Grouping" table on the Size/Cost Assignments panel, as well as the headers used in the "Assign Candidate Sets to Pipes" section on the [Candidate Sets panel](#) and the "Assign Design Requirements to Pipes" section on the [Design Requirements panel](#).

Sizing Navigation Panel Preferences

This section allows the user to set the default location for the Sizing Navigation Panel. By default the Sizing Navigation Panel will be located at the bottom of the Sizing window.

Sizing Panel Preferences

This section provides users with the option to show the discrete/continuous option on the [Candidate Sets panel](#). By default this option will be hidden and all Candidate Sets will be defined as discrete.

In general the continuous candidate set option should only be used if the Sizing Type is set to Continuous, and [discrete sizing](#) does not need to be performed for the pipes.

Related Topics

User Options Overview.....	446
Unit System.....	447
Workspace.....	449
Parameters.....	452

Files.....454

File Import and Export

This section focuses on the various file types that can be imported and exported from AFT Fathom. Topics included discuss importing piping layouts from various sources, importing and exporting between AFT products, and importing and exporting data with excel.

Contents

Importing Piping Component Files	458
Importing GIS Shapefiles	462
Importing CAESAR II Neutral Files	464
Importing and Exporting EPANET Files	467
Importing and Exporting from Other AFT Products	471
Exporting To Excel	472
Importing From Excel	478

Importing Piping Component Files

AFT Fathom includes the ability to import Piping Component Files (.pcf), which can be generated by a wide range of software, and provide a quick way to get started with the hydraulic modeling portion of a project. Files can be imported into existing models.

Selecting File > Import Piping Layout From > Piping Component File (.pcf) brings up the Import Piping Layout Wizard in the [Quick Access Panel](#), as shown below in Figure 1.

The Import Piping Layout Wizard is organized into sections that help guide the import process:

- **Folder Browser** - After browsing to a folder, all .pcf files within the folder will be listed here. Simply clicking a file in the list will show a preview of the import in the [Workspace](#). The location of the preview can be moved by dragging the icon in the upper-left of the selection area.
- **Object List** - When a file is selected, the objects to be created will be listed here. Each individual object can be reassigned to a certain [Junction](#), if the automatic assignment is not correct. For example, the originating software may have only had a "valve" while this could be a [Relief Valve](#) or a [Control Valve](#). Changing the AFT Object type will change the preview in the Workspace. Right-clicking on an object shows a menu for converting object types.
 - **Property Window** - This button opens an additional panel that shows the raw properties for the currently selected object.
 - **Link Materials** - Opens the File Object Link window, where groups of pipes from the .pcf file can be associated with a pipe material, size, and type in the AFT Pipe Material library.

Pipes can be grouped by file schedules (i.e. item codes) or diameters, and are sorted by length within each group. The selection from the AFT Pipe Material library will overwrite any diameter information from the .pcf file. Selections made for a .pcf material will be applied to all pipes that reference that .pcf material. Pipes associated with each material can be seen by clicking the arrow on the left expand the group (Figure 2) and can be edited individually or as a group.

- **Show Ignored Objects** - Certain types of objects are ignored on import - these can be objects such as flanges, or additional file components such as materials or bolt specifications.
- **Options**
 - **Min Pipe/Max Pipe** - Information fields on the length of the shortest pipe and longest pipe in the list of pipes to be imported.
 - **Ignore Pipes of Length Less Than** - Do not import pipes shorter than this amount.
 - **Ignore Pipes of Diameter Less Than** - Do not import pipes smaller than this diameter.
 - **Ignored Pipes** - How many pipes will be ignored based on the above settings. Ignoring pipes can be helpful in keeping a model clean and free of unnecessary components. For example, a pipe stress software may require many short pipe segments in between welds or flanges. However, these types of connections are almost always ignored for hydraulic analysis. By ignoring short pipes, the amount of "import clean up" required can be reduced.
 - **Preferred Scale** - The wizard will **Auto Scale** the import by default. If a different scale is desired, this option can be disabled and a custom scale entered.
 - **Show Pipes in Preview** - Display the pipes in the Workspace preview.
 - **Show Junctions in Preview** - Display the junctions in the Workspace preview.
 - **Combine Pipes When Possible** - It is required that all pipes in the Workspace be connected to other pipes by a junction. However, this is not a requirement in all software. Therefore, some .pcf files may be generated with several pipes "end to end." If the pipes were imported this way, either junctions would need to be added, or the pipes would need to be combined before the model could be solved. This option attempts to automatically combine pipes in this situation.
 - **Import Bends as Pipes** - Pipe bends can be imported as junctions, or as a visual change in direction of the pipe. Importing bends as pipes is identical to using [segments](#) with assigned fittings and losses to show piping layout. To account for losses, bends will be matched to smooth flanged bends in the AFT Internal [Fittings & Losses Library](#) if an exact match for angle and r/D ratio is available and assigned to the pipe automatically. Losses that are not assigned automatically can be set up manually using junctions or [fittings & losses](#). This option is helpful to reduce unnecessary model clutter and input **if bend losses are negligible**.

Note: In most cases, it is recommended to import bends as bends rather than pipes. Bends imported as bends will always use smooth bends with r/D and/or angle imported from the .pcf file and basis area inherited from the upstream pipe.

- **Ignore Fitting Lengths** - Neglects additional length added to pipes that accounts for the equivalent length of fittings and losses, and is not actually part of the physical pipe length.
- **Import Data to Pipe or Junction Notes Field** - Add the raw data from the originating file to the [Notes](#) tab of the corresponding object.

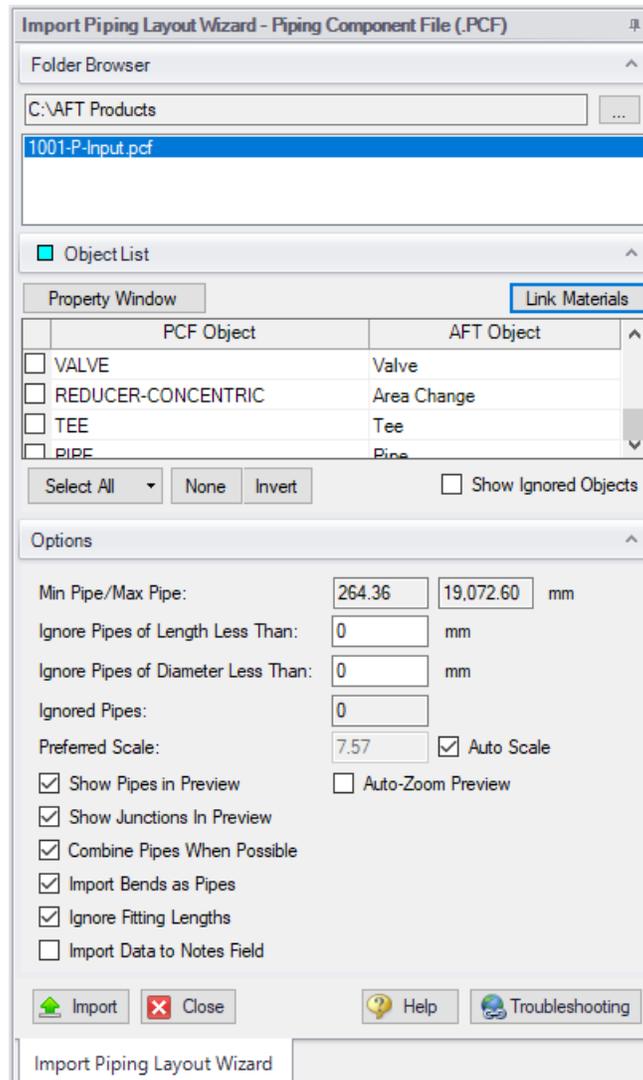


Figure 1: Piping Component File Import

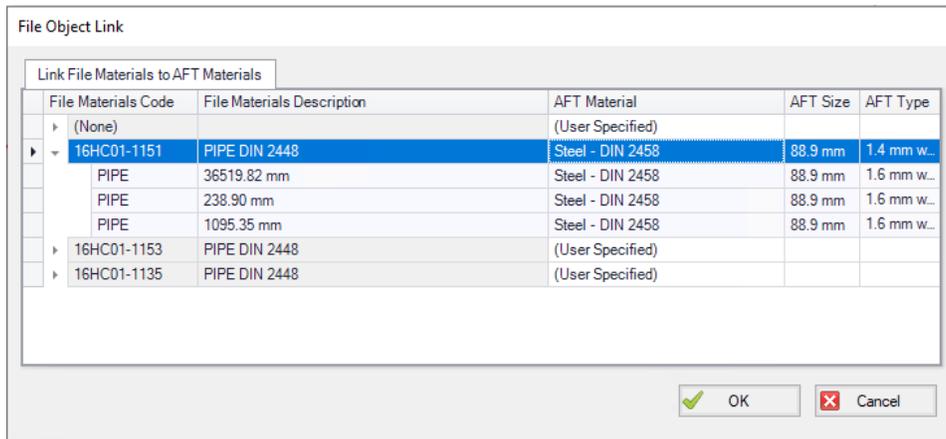


Figure 2: File Object Link window for Piping Component File Import

General Recommendations

Importing piping layouts from external file types such as Piping Component Files is intended to save time by allowing a means to quickly achieve a foundational layout. A .pcf file does not provide all information needed to create a working hydraulic model of a system and the import process will not produce a completed model or a perfectly accurate representation of the .pcf model. There will be missing data that must be assigned prior to analysis, such as fluid properties and boundary conditions. However, there are import practices and preparations that can be made to the .pcf file that will make creating model from an imported piping layout go more smoothly.

- **Workspace** - Piping Component Files are a three-dimensional file format, whereas model layouts in the AFT Workspace are a two-dimensional representation of a system. Because the Workspace is a purely visual tool independent of actual system dimensions, the [Scale/Flip Workspace](#) feature can be useful for achieving a readable model when working with imported piping layouts.
- **Object Positioning** - It is recommended to have some distance between most objects (this is not relevant to welds, branch/tap connections, gaskets, and supports). This aids in pipe connections, as objects that are very close together may have trouble connecting automatically. This is especially true of non-isometric models. Spacing between objects also streamlines the process of scaling objects on the Workspace after import.
- **Bends** - By default, Import Bends as Pipes is unchecked, and importing bends as bends is recommended for the sake of model readability and a more flexible approach to incorporating bend losses. Bends that have common angles and r/D ratios present in the internal AFT library can be imported as [Fittings & Losses](#) automatically when importing bends as pipes, but other bends will need to be added manually as junctions, or have their losses specified using the Additional K Factor option in the Pipe Fittings & Losses window or as user-specified fittings & losses using the [Fittings & Losses Library](#). When bends are imported as bends, they will be treated as smooth bends, the upstream pipe determines the cross-sectional flow area, and the r/D ratio and/or angle is imported from the Piping Component File. If only the r/D ratio or angle is specified, the other parameter will be calculated using the endpoint coordinates.

Note: Bends modeled explicitly can be joined with adjacent pipes as using [Join Pipes/Junctions](#) later if desired. Modeling bends as explicit junctions is advised for models containing bends that are not smooth flanged or have an uncommon angle or r/D ratio.

- **Pipes** - While the AFT Pipe Material Library contains many piping materials specified by ASME and ANSI, the Piping Component File format does not always contain these materials and material attributes can be defined within .pcf files in a number of drastically different ways. Because of this, materials, nominal sizes and schedules are not assigned automatically and can be assigned using Link Materials (shown above in Figure 2). Pipes can be grouped by file schedules (i.e. item codes) if they exist or by diameter so that pipe size and material can be applied to a group of pipes quickly. Pipe inner diameter is assigned from the nominal diameter defined in the pipe's coordinates within the .pcf file as a starting point, but any sizes assigned in Link Materials will override these inner diameters within the AFT model. Alternatively, pipes can be defined after import using [Global Pipe Edit](#).

Related Topics

File Import and Export	458
Importing GIS Shapefiles	462
Importing CAESAR II Neutral Files	464
Importing and Exporting EPANET Files	467
Importing and Exporting from Other AFT Products	471
Exporting To Excel	472
Importing From Excel	478

Importing GIS Shapefiles

Import GIS Shapefile

GIS Shapefiles that adhere to the ESRI standard can be imported into AFT Fathom for ease of model creation. These files have the extension .shp, but there are two other supporting file types that need to be in the same folder as the .shp in order for the import to work correctly. These are .shx and .dbf file types. The option is located under File > Import Piping Layout From > GIS Shapefile (.shp). This opens the "Import Piping Layout Wizard" in the [Quick Access Panel](#) that allows the importing of one line .shp that is of pipes and multiple point .shp files for branches, valves, pumps, and reservoirs. Once the .shp files have been added to this window the object type, scaling, and mapping can be selected.

Object Type

Any .shp files that are of lines are automatically set to the object type of "Pipes". Only one .shp file containing lines can be imported at a time. Any .shp file that is of points can be set to Branches, Valves, Pumps, or Reservoirs.

Scaling

Scaling only applies when no length is mapped to the AFT "length" field or when no length data exists in the shapefile. Scaling does not alter the Workspace layout or visual length of the pipe. It only adjusts the pipe length property that is calculated by the coordinates being imported from the shapefile.

The default scaling is 1 grid point to 1 foot. This can be adjusted by selecting the "... " button in the Scaling column. This button opens a new window in which a new scale is applied to the automatic calculation of the pipe length property. This window does not change the visual length of the pipe drawn on the workspace. This window also allows the viewing of the coordinates of specific points and parts of different shapes within the shapefile.

Mapping

The Data Mapping window allows the user to select what data fields have relevant data for AFT Fathom to import.

- Pipes can map:
 - **Labels** - Names
 - **Length** - Mapping this field will utilize included user shapefile data and override the automatic calculation of pipe length. The scaling window settings will have no effect when this field is mapped.
 - **Friction** - Absolute Roughness, Relative Roughness, Hazen-Williams Factor, Explicit Friction Factor, Weymouth, and Panhandle friction models are available.
 - **Inner Diameter**
- Branches can map:
 - **Labels** - Names
 - **Elevation**
- Pumps can map:
 - **Labels** - Names
 - **Elevation**
 - **Assigned Flow** - Imports as a Fixed Flowrate pump
- Reservoirs can map:
 - **Labels** - Names
 - **Elevation** - To the base of the reservoir
 - **Liquid Elevation** - Height from base to liquid surface elevation. Liquid elevation and elevation are summed and input as liquid surface elevation in the generated reservoir junctions.
- Valves can map:
 - **Labels** - Names
 - **Elevation**

Related Topics

File Import and Export.....	458
Importing Piping Component Files.....	458
Importing CAESAR II Neutral Files.....	464
Importing and Exporting EPANET Files.....	467
Importing and Exporting from Other AFT Products.....	471
Exporting To Excel.....	472
Importing From Excel.....	478

Importing CAESAR II Neutral Files

AFT Fathom includes the ability to import CAESAR II Neutral Files (.cii), providing a quick way to get started with the hydraulic modeling portion of a project. Files can be imported into existing models.

Selecting File > Import Piping Layout From > CAESAR II Neutral Files (.cii) brings up the Import Piping Layout Wizard in the [Quick Access Panel](#), as shown below in Figure 1.

The Import Piping Layout Wizard is organized into sections that help guide the import process:

- **Folder Browser** - After browsing to a folder, all .cii files within the folder will be listed here. Simply clicking a file in the list will show a preview of the import in the [Workspace](#). The location of the preview can be moved by dragging the icon in the upper-left of the selection area.
- **Object List** - When a file is selected, the objects to be created will be listed here. Each individual object can be reassigned to a certain [Junction](#), if the automatic assignment is not correct. For example, the originating software may have only had a "valve" while this could be a [Relief Valve](#) or a [Control Valve](#). Changing the AFT Object type will change the preview in the Workspace. Right-clicking on an object shows a menu for converting object types.
 - **Property Window** - This button opens an additional panel that shows the raw properties for the currently selected object.
 - **Link Materials** - Opens the File Object Link window, where the pipes can be associated with a pipe material, size, and type in the AFT Pipe Material library. The selection from the AFT Pipe Material library will overwrite any diameter information from the CII file, and will be applied to all of the pipes. Pipes are grouped by diameter by default and can also be grouped by schedule.
 - **CAESAR II Info** - Detailed information from the neutral file can be viewed for objects in the neutral file, including information on physical properties, allowable stresses, 3-dimensional orientation, etc. for the imported objects.
- **Options**
 - **Min Pipe/Max Pipe** - Information fields on the length of the shortest pipe and longest pipe in the list of pipes to be imported.
 - **Ignore Pipes of Length Less Than** - Do not import pipes shorter than this amount.
 - **Ignore Pipes of Diameter Less Than** - Do not import pipes smaller than this diameter.
 - **Ignored Pipes** - How many pipes will be ignored based on the above settings. Ignoring pipes can be helpful in keeping a model clean and free of unnecessary components. For example, a pipe stress software may require many short pipe segments in between welds

or flanges. However, these types of connections are almost always ignored for hydraulic analysis. By ignoring short pipes, the amount of "import clean up" required can be reduced.

- **Preferred Scale** - The wizard will **Auto Scale** the import by default. If a different scale is desired, this option can be disabled and a custom scale entered.
- **Show Pipes in Preview** - Display the pipes in the Workspace preview.
- **Show Junctions in Preview** - Display the junctions in the Workspace preview.
- **Show Labels In Preview** - Show the corresponding label from the .cii file in the Workspace preview. This is only shown in the preview and will not be imported into the model itself.
- **Combine Pipes When Possible** - It is required that all pipes in the Workspace be connected to other pipes by a junction. However, this is not a requirement in all software. Therefore, some .pcf files may be generated with several pipes "end to end." If the pipes were imported this way, either junctions would need to be added, or the pipes would need to be combined before the model could be solved. This option attempts to automatically combine pipes in this situation.
- **Import Bends as Pipes** - Pipe bends can be imported as junctions, or simply as a visual change in direction of the pipe. Importing bends as pipes is identical to using [segments](#) to show piping layout - no additional losses due to direction change are accounted for with this option. To account for losses, additional information will need to be entered using either junctions or [fittings & losses](#). This option is helpful to reduce unnecessary model clutter and input if bend losses are negligible.
- **Ignore Fitting Lengths** - Neglects additional length added to pipes that accounts for the equivalent length of fittings and losses, and is not actually part of the physical pipe length.
- **Import Data to Pipe or Junction Notes Field** - Add the raw data from the originating file to the [Notes](#) tab of the corresponding object.

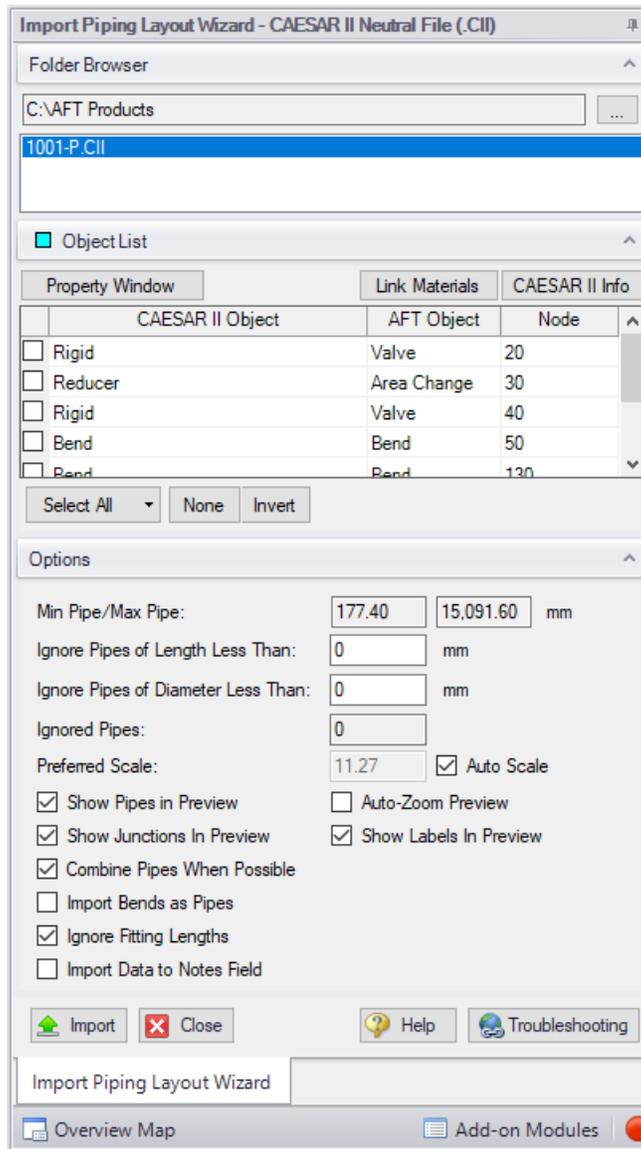


Figure 1: CAESAR II Neutral File Import

Related Topics

File Import and Export.....	458
Importing Piping Component Files.....	458
Importing GIS Shapefiles.....	462
Importing and Exporting EPANET Files.....	467
Importing and Exporting from Other AFT Products.....	471
Exporting To Excel.....	472
Importing From Excel.....	478

Importing and Exporting EPANET Files

AFT Fathom includes the ability to import and export EPANET Files (*.INP). EPANET is public domain modeling software developed by the US Environmental Protection Agency. Many different applications support the file format, so it can therefore provide a quick way to get started with the hydraulic modeling portion of a project.

Import Piping Layout Wizard

Selecting File > Import Piping Layout From > EPANET File (.inp) brings up the Import Piping Layout Wizard in the [Quick Access panel](#), as shown below in Figure 1. Note that importing from EPANET requires that the Workspace be empty.

The Import Piping Layout Wizard is organized into sections that help guide the import process:

- **Folder Browser** - After browsing to a folder, all .inp files within the folder will be listed here. Simply clicking a file in the list will show a preview of the import in the [Workspace](#). The location of the preview can be moved by dragging the icon in the upper-left of the selection area.
- **Object List** - When a file is selected, the objects to be created will be listed here. Each individual object can be reassigned to a certain [Junction](#), if the automatic assignment is not correct. For example, the originating software may have only had a "valve" while this could be a [Relief Valve](#) or a [Control Valve](#). Changing the AFT Object type will change the preview in the Workspace. Clicking on an object in the workspace highlights it in the wizard.
 - **Property Window** - This button opens an additional panel that shows the raw properties for the currently selected object.
 - **Link Materials** - Opens the File Object Link window, where the pipes can be associated with a pipe material, size, and type in the AFT Pipe Material library. The selection from the AFT Pipe Material library will overwrite any diameter information from the EPANET file, and will be applied to all of the pipes.
- **Options**
 - **Min Pipe/Max Pipe** - Information fields on the length of the shortest pipe and longest pipe in the list of pipes to be imported.
 - **Ignore Pipes of Length Less Than** - Do not import pipes shorter than this amount.
 - **Ignore Pipes of Diameter Less Than** - Do not import pipes smaller than this diameter.
 - **Ignored Pipes** - How many pipes will be ignored based on the above settings. Ignoring pipes can be helpful in keeping a model clean and free of unnecessary components. For example, a pipe stress software may require many short pipe segments in between welds or flanges. However, these types of connections are almost always ignored for hydraulic analysis. By ignoring short pipes, the amount of "import clean up" required can be reduced.
 - **Preferred Scale** - The wizard will **Auto Scale** the import by default. If a different scale is desired, this option can be disabled and a custom scale entered.
 - **Show Pipes in Preview** - Display the pipes in the Workspace preview.
 - **Show Junctions in Preview** - Display the junctions in the Workspace preview.

- **Import Data to Notes Field** - Add the raw data from the originating file to the [Notes](#) tab of the corresponding object.
- **Auto-Zoom Preview** - This option can be enabled to automatically zoom out so that the whole model can be seen in the workspace, which can be convenient for large systems.

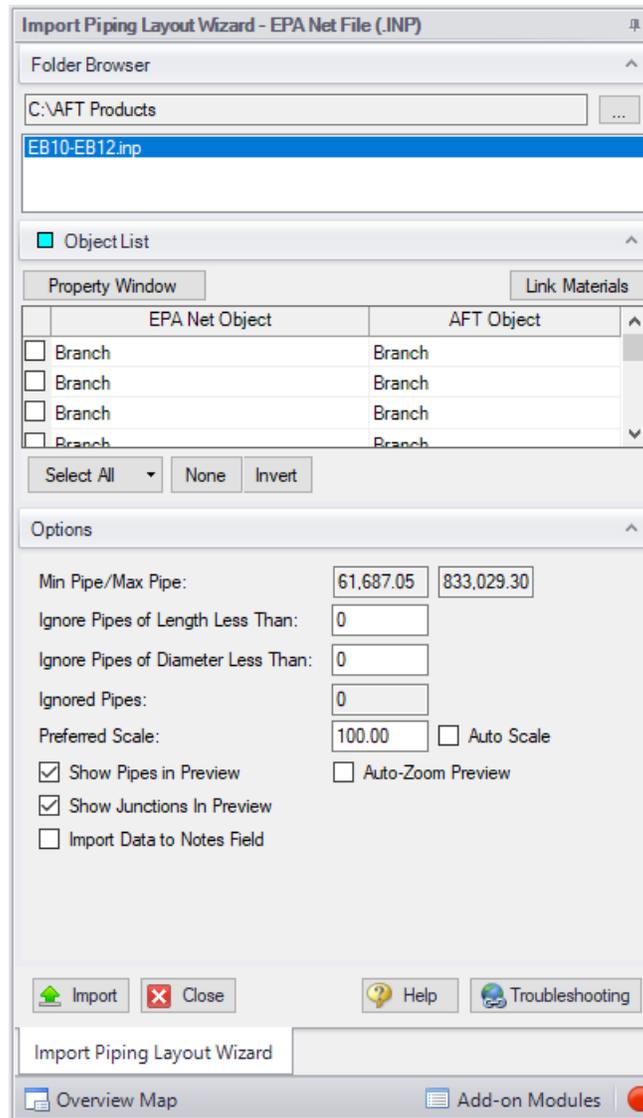


Figure 1: EPANET Import/Export Dialogs

Pipes and Junctions

All pipes and junctions get imported with user-specified names from EPANET. All junctions below that have flow, elevation, length or other parameters will be imported with the same units if possible.

- **Pumps** - The available points on the pump curve are imported into the pump data and used with the curve fit type set to interpolated X-Y data. If desired, polynomial curve fits can be set up within pump configuration using this data after the import process is complete.

Note: If only one or two data points exist for the pump curve, the curve will still need to be user-specified.

- **Tanks** - If XTS is on, finite-volume tanks will be imported as finite open tanks. Otherwise, they will be imported as normal reservoirs.
- **Reservoirs** - EPANET reservoirs are imported as a reservoir with a surface pressure of 1 atm with liquid elevation imported from EPANET.
- **Pipes** - These will use roughness, length, and inner diameter from EPANET model.
- **Junctions** - (Just called Junctions in EPANET)
 - **If one connecting pipe** - If it has a demand, it becomes an Assigned Flow with relevant inflow/outflow. Otherwise, it becomes a dead end.
 - **If multiple connecting pipes** - If it has a demand, it becomes a branch using the Flow Source or Sink optional property with the relevant inflow/outflow imported from the EPANET junction demand. Otherwise, it is imported as a normal branch.
- **Valves** - The K factor of a valve is assigned with the value specified in the EPANET model. Valves can be imported as normal valves, control valves, relief valves, or check valves depending on how they are defined within EPANET:
 - Normal valves come from GPV and TCV in EPANET.
 - Control valves come from FCV, PBV, and PSV in EPANET.
 - Relief valves come from PRV in EPANET.
 - Check valves come from CV in EPANET.
- **Heat Exchangers** - While Heat Exchangers are not an explicitly defined object type in EPANET, EPANET junctions with "Heat Exchanger" in their user-defined name are imported as Heat Exchangers in AFT software.
- **Emitters** - Emitters in EPANET are defined as a junction with a base demand and an emitter coefficient. These are imported as Spray Discharge junctions with the emitter coefficient imported as a K Fire Sprinkler value with the flow units defined by the base demand units and the pressure component defined by the emitter coefficient units (e.g. flow unit per square root of psi in EPANET). The number of sprinklers per emitter are set to 1 and the exit pressure is set to the atmospheric pressure (zero gauge pressure). The correct atmospheric pressure for the imported system should be set in Environmental Settings in Analysis Setup.

Exporting EPANET Files

AFT models can be exported to an EPANET file by using the File -> Export Piping Layout To -> EPANET File (.inp)... option from the Workspace window. The EPANET Export window shown in Figure 2 below allows the user to specify the desired destination to export the file to.

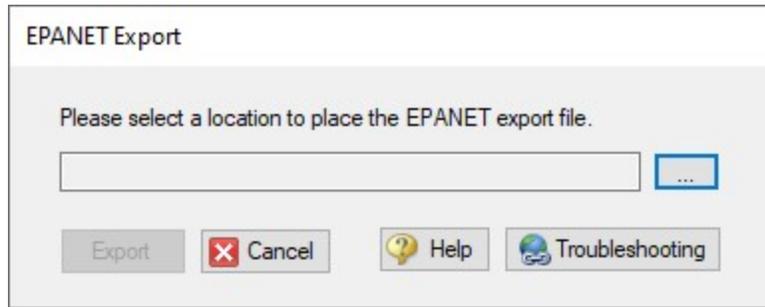


Figure 2: EPANET Import/Export Dialogs

Related Topics

File Import and Export.....	458
Importing Piping Component Files.....	458
Importing GIS Shapefiles.....	462
Importing CAESAR II Neutral Files.....	464
Importing and Exporting from Other AFT Products.....	471
Exporting To Excel.....	472
Importing From Excel.....	478

Importing and Exporting from Other AFT Products

Models made in other AFT products such as AFT Impulse can be opened directly in AFT Fathom and vice versa. This can be a useful starting point when it is desired to perform additional analysis on a model that has already been built in another AFT product, such as if it is desired to plot pump vs system curves for a model built in Impulse.

To convert a model from one AFT product to another just browse to the model you would like to convert and open it in the software you are converting the model to. This can be done by changing the file extension selected, as is shown below in Figure 1. Note that though AFT Arrow and AFT xStream are not explicitly listed, models from Arrow and xStream can also be converted by selecting "All" and browsing to the model file.

When the model is opened in AFT Fathom a backup copy of the original model will be created from which the original model can be restored if needed. See the [Backup Settings](#) explanation for more information on backup files.

The conversion process will automatically change junction types not included in AFT Fathom to either a General Component or Branch junction. Be aware that some features available for use in other AFT products are not available for use in AFT Fathom. If a junction is defined using a feature not available in AFT Fathom, the non-applicable information will be removed, and the user may be required to input additional information to fully define the model. It is advised to fully review the model after import.

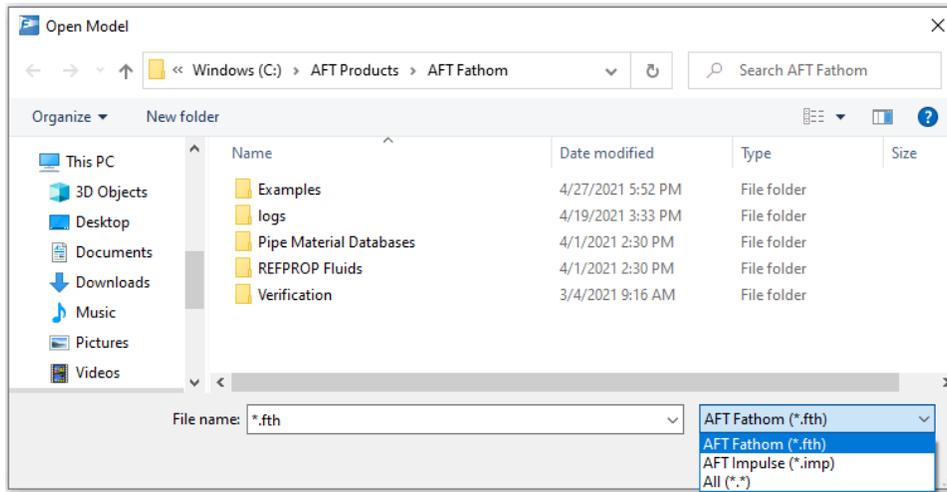


Figure 1: File extensions options for Open File window

Related Topics

File Import and Export	458
Importing Piping Component Files	458
Importing GIS Shapefiles	462
Importing CAESAR II Neutral Files	464
Importing and Exporting EPANET Files	467
Exporting To Excel	472
Importing From Excel	478

Exporting To Excel

AFT Fathom includes a powerful ability to export any [Output](#) data to any specific cell(s) in an Excel spreadsheet. Specific values from the model can be exported, as well as columns, rows, or entire output tables as defined in [Output Control](#). These items are controlled with the Excel Export Manager, available from the File menu.

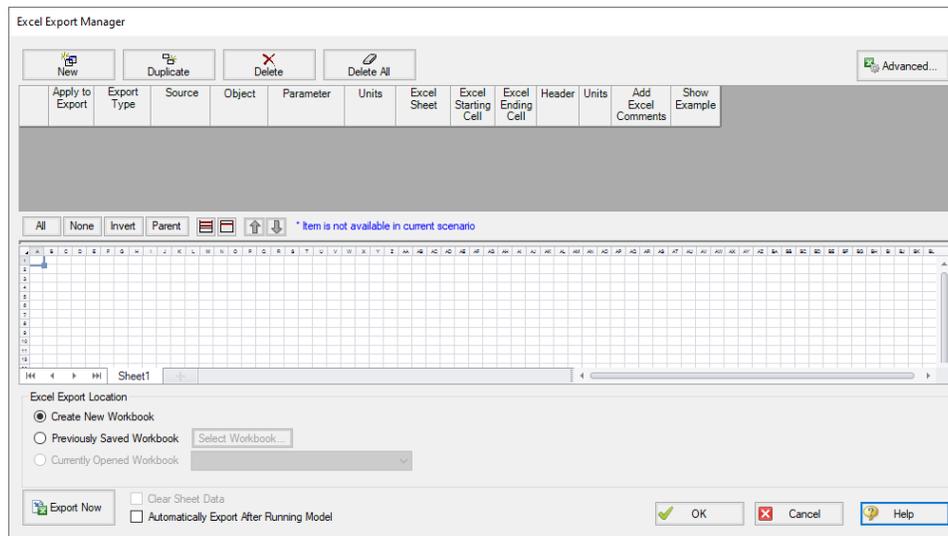


Figure 1: The Excel Export Manager

Adding Items to the Excel Export Manager

Data from the Output can be added to the Excel Export by clicking New, or by selecting an existing item and clicking Duplicate. Items can also be created directly from the [Output Window](#) by right-clicking on a cell or header.

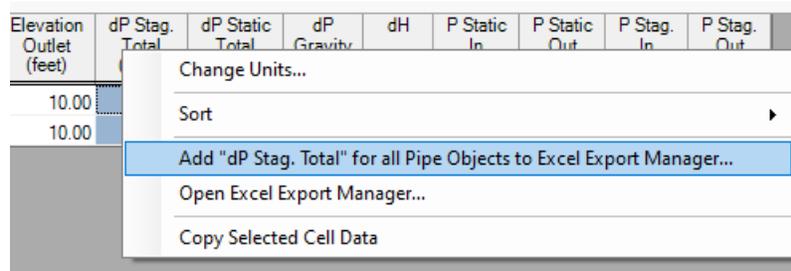


Figure 2: Adding items directly from the Output

Each row in the Excel Export Manager corresponds to a cell, column, row, or table of data to be added to the spreadsheet. Each column helps define what data will be transferred where. The grid below the entries shows a preview of where the data will be added to help prevent intersecting data by showing where there are overlapping regions. Note that graph items can vary in size and do not show a complete region here.

- **Apply to Export** - Items can be optionally included or excluded from the export.
- **Export Type** - What type of data is being exported. This affects other settings, and will affect how much room is required on the target Excel sheet.

- **Single Value** - The value of a single parameter for a single pipe or junction object.
 - **Column** - Values of a single parameter for all of the [displayed](#) objects.
 - **Row** - Values of all [displayed](#) parameters for a single object.
 - **Table** - Values of all [displayed](#) parameters for all [displayed](#) objects.
 - **Graph** - The graph points from a specified [Graph List Item](#).
 - **Other** - Additional information not available in a standard Output table, such as Fluid definition, Model Title, or Scenario Names.
- **Source** - Where the data is coming from.
 - **Pipe** - From the Pipe section of the output.
 - **Junction** - From the Junction section of the output.
 - **General Output Summary** - From one of the available output Summary sections. For example, if there is a pump junction in the model, Pump Summary is available.
 - **Graph** - From a specified [Graph List Item](#).
 - **Object** - The object in particular that is desired. If Column or Table are selected for Export Type, this selection will be fixed to ALL.
 - **Parameter** - The parameter in particular that is desired. If Row, Table or Graph are selected for Export Type, this selection will be fixed to ALL.
 - **Units** - The units of the exported value. This is available only for Single Value and Column Export Types. Some parameters do not have units and this will display N/A.
 - **Excel Sheet** - Which Excel Sheet to export to. This value can be edited by the user - if it is desired to export to a named sheet this can be entered here.
 - **Excel Starting Cell** - The upper-left cell to export the data to.
 - **Excel Ending Cell** - The automatically calculated lower-right cell of the data, which depends on the Export Type, export options, and the number of parameters or objects in the model. It is important to recognize that the Excel Starting Cells can be defined in a way that data gets **overwritten**. The Ending Cell column is a handy reference to help avoid this.
 - **Header** - Optionally export the name of the Parameter(s) related to the item.
 - **Units** - Optionally export the name of the Unit(s) related to the item.
 - **Add Excel Comments** - Optionally adds an Excel comment to the value cell that contains the Parameter name and unit.
 - **Show Example** - Displays an example of how the exported cells will appear.

Excel Export Manager													
Apply to Export		Export Type	Source	Object	Parameter	Units	Excel Sheet	Excel Starting Cell	Excel Ending Cell	Header	Units	Add Excel Comments	Show Example
<input checked="" type="checkbox"/>	Row	Pipe	P104 (Supply 1)	ALL		ALL	Sheet1	A1	O3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	Row	Pipe	P110 (CT 1 Hdr A)	ALL		ALL	Sheet1	B5	O6	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	Column	Pipe	ALL	Volumetric Flow Rate	gal/min		Sheet1	A9	B43	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	Table	Pump Summary	ALL	ALL		ALL	Sheet1	D9	Q14	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	Other	N/A	N/A	Scenario Name	None		Sheet1	D16	D16	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	

Figure 3: Example Export

	A	B	C	D	E	F	G	H
1		Name	Vol. Flow Rate	Velocity	Pipe Nominal Size	dP Static Friction Wall	P Static Max	P Static Min
2			gal/min	feet/sec		psid	psig	psig
3	P102 (Supply 1)	Supply 1	16064.8359	7.8032	30 inch	2.008	26.3535	24.0517
4			gal/min	feet/sec		psid	psig	psig
5		CT 1 Hdr A	10628.3047	8.0317	24 inch	0.1086	12.758	12.6495
6								
7								
8								
9		Vol. Flow Rate		Name	Vol. Flow	Mass Flow	dP	
10	P100 (Circ Pump 1A Suction)	8031.9014			gal/min	lbm/sec	psid	
11	P101 (Circ Pump 1A Disch)	8031.9014		J101 (Cooling Tower 1 Circ Pump A)	8031.9023	1110.399	24.5529	
12	P102 (Supply 1)	16064.8359		J102 (Cooling Tower 1 Circ Pump B)	8031.9023	1110.399	24.5529	
13	P103 (LO Clr 1 In)	830.5782		J201 (Cooling Tower 2 Circ Pump A)	8031.9023	1110.399	24.5529	
14	P106 (LO Clr 1 Out)	834.2666		J202 (Cooling Tower 2 Circ Pump B)	8031.9023	1110.399	24.5529	
15	P107 (Cond 1 In)	15233.8105						
16	P108 (Cond 1 Out)	15301.46		Scenario: Base Scenario				
17	P109 (Return 1)	16136.1387						

Figure 4: Result of the above example, color added for emphasis

Excel Export Location

After all of the desired data has been added to the Excel Export Manager, the next step is to actually export the data. There are three options available for the export:

- **Create New Workbook** - A new Excel Workbook will be opened with the exported data.
- **Previously Saved Workbook** - Browse to a Workbook saved to file. The Workbook will be opened and the modifications made to the sheets as specified.
- **Currently Opened Workbook** - Select a Workbook that is currently open in Excel. The already open Workbook will be modified.

Note: The Excel Export Manager **does not** save Excel files for non-batch run exports. Any changes made need to be manually saved by the user in Excel.

There are two options available for exporting:

- **Clear Sheet Data** - All of the existing data on the sheets specified in the Excel Sheet column will be cleared. This is especially useful for setting up a new export when many changes are being made.
- **Automatically Export After Running Model** - After the model is run, the Export will be carried out automatically, per the settings defined by the user in the Excel Export Manager.

Advanced Options

Items can be added directly from the Output instead of adding custom items with the New button. There are two behavioral options that can be modified for this action:

- **Open Excel Export Manager** - If enabled, the Excel Export Manager will be opened every time an item is added from the Output. This is useful for double checking the export process completed as expected.
- **For Existing Items** -

- **Add New Item** - The item selected will be added to the Excel Export Manager even if it already exists.
- **Select Existing** - If the item already exists, do not add a duplicate item and instead select the existing item.
- **Ask** - If the item already exists, prompt the user on the action to take.
- **Create an Export Guide** - When exporting, an Export Guide can optionally be included in the export. The Export Guide contains information about the model and the generation of the export, to help keep the connection between the data and the model clear for future reference.

Exporting Data From Multiple Scenarios

The Excel Export Manager can be employed with [Batch Runs](#) with a couple of important distinctions.

First, if multiple sheets are specified in the Excel Export Manager, the only way to export multiple scenarios is to export **every scenario to its own Workbook**. To keep data from multiple scenarios in one Workbook, there can only be one sheet specified in the Excel Export Manager.

Second, **the Excel File will be saved after each scenario**. In normal single-scenario exports the file is never saved, but in batch mode it is required.

In the Batch Run tool, users are also able to use Excel Export without having to run the scenarios again by using the Export Only (Do Not Run) option. Note that if any of the scenarios in the Batch Run do not have output, the batch Export Only will not export any data for that scenario, as it will not run those scenarios to get output.

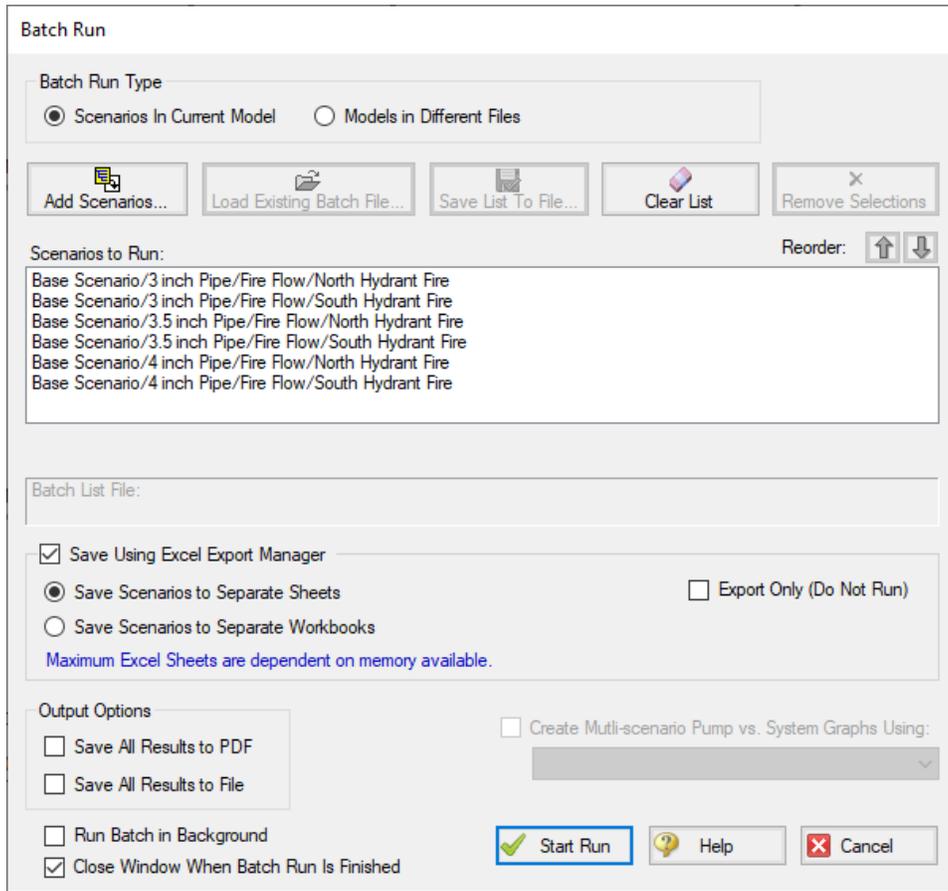


Figure 5: Saving Results From Multiple Scenarios

Related Examples

Flow Induced Vibration Calculations (Metric Only).....985

Related Topics

File Import and Export.....458
 Importing Piping Component Files.....458
 Importing GIS Shapefiles.....462
 Importing CAESAR II Neutral Files.....464
 Importing and Exporting EPANET Files.....467
 Importing and Exporting from Other AFT Products.....471
 Importing From Excel.....478

Importing From Excel

A Microsoft Excel spreadsheet can be used to make changes to objects in an AFT Fathom model. The values in the Excel file are located in specific locations as described in the Setting Up Excel section.

Setting Up Excel

When reading an Excel file, AFT Fathom will search for a worksheet named **AFT Transfer**. There can be other worksheets in the workbook but Fathom will only look for a sheet with this name. The file structure looks like the following:

	A	B	C	D	E	F	G	H
1	Comment	Apply	Object Type	Object Number	Parameter	Change Code	Value	Scenario Path Name
2	HEX heat rate set	Yes	Heat Exchanger	67	Heat Rate	Set equal to value	4000	High Heat
3	HEX heat rate set	Yes	Heat Exchanger	66	Heat Rate	Set equal to value	3200	Low Heat
4	HEX heat rate set	Yes	Heat Exchanger	65	Heat Rate	Set equal to value	3700	Normal Heat
5	FCV setpoint set	Yes	Control Valve	71	Control Setting	Set equal to value	300	
6	FCV setpoint reduce by 20 gpm	Yes	Control Valve	72	Control Setting	Change by value (+/-)	-20	
7	FCV setpoint increase by 10%	Yes	Control Valve	73	Control Setting	Change by percent (+/-)	10	
8	Pump Speed set	Yes	Pump	59	Pump Speed (%)	Set equal to value	90	
9								

Figure 1: Sample AFT Transfer Excel Sheet

Row 1 and column A are not read by AFT Fathom - these cells can have any value in them, though it is recommended to enter the values above for the header row.

Each column has certain accepted values, which can depend on the values in neighboring columns.

Note: A template spreadsheet containing validation rules for the required layout is included in the AFT Fathom installation, or can be downloaded [here](#). By default the file path is: C:\AFT Product-s\AFT Fathom\AFT Fathom Excel Import Workbook.xlsx

Apply Column

This column accepts Yes or No. Only changes marked as Yes will be imported into the model.

Object Type

This column must be a junction name or Pipe. Valid choices can be seen below in Table 1.

Object Number

This corresponds to the junction or pipe number.

Parameter

The specific parameter to be changed. These parameters vary based on the Object Name selected. Each parameter has a type (C/S/I) associated with it, as is explained in the next section.

Table 1: Object Types, Associated Parameters, and Change Codes

Pipe	Area Change	Assigned Flow	Assigned Pressure	Bend
S Additional K Factor	S Design Factor	S Design Factor	S Design Factor	S Design Factor
S Ambient Fluid Velocity	S Initial Guess Pressure	S Elevation	S Elevation	S Initial Guess Pressure
S Ambient Temperature	S Initial Guess Temperature	S Flow Rate	C Junction Name	S Initial Guess Temperature
S Buried Pipe Depth	S Inlet Elevation	S Initial Guess Pressure	S Pressure	S Inlet Elevation
S Convection Coefficient External	C Junction Name	C Junction Name	I Sizing/Cost Setting	C Junction Name
S Convection Coefficient Internal	S Outlet Elevation	I Sizing/Cost Setting	S Temperature	S Outlet Elevation
S Cylindrical Annulus Inner Pipe OD	I Sizing/Cost Setting	I Special Condition		I Sizing/Cost Setting
S Cylindrical Annulus Outer Pipe OD	S User Specified K	S Temperature		S User Specified K
S Design Factor - Pipe Fittings & Losses				
S Design Factor - Pipe Friction				
S Design Factor - Pipe Heat Transfer				
S External Heat Flux/Tracing Flux				
S External Heat Rate				
S ID Reduction (Scaling)				

Interface and User Options

<p>S Initial Guess Flowrate</p> <p>S Inner Diameter (for unspecified, cylindrical pipes)</p> <p>S Length</p> <p>I Nominal Size Increment</p> <p>C Nominal Size Set (must match text in Pipe Property Window)</p> <p>C Nominal Type Set (must match text in Pipe Property Window)</p> <p>S Noncylindrical Pipe Flow Area</p> <p>S Noncylindrical Pipe Wetted Perimeter</p> <p>C Pipe Name</p> <p>S Rectangular Duct Height</p> <p>S Rectangular Duct Width</p> <p>S Roughness Value (for unspecified friction models)</p> <p>I Sizing/Cost Setting</p> <p>S Soil Surface Temperature</p> <p>I Special Condition</p>				
<p>Branch</p> <p>S Design Factor</p>	<p>Check Valve</p>	<p>Control Valve</p>	<p>Dead End</p> <p>S Initial Guess</p>	<p>General Component</p>

Interface and User Options

<p>S Elevation</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>C Junction Name</p> <p>I Sizing/Cost Setting</p> <p>S Source/Sink Flow Rate</p> <p>S Source/Sink Temperature</p> <p>I Special Condition</p>	<p>S Design Factor</p> <p>S Forward Velocity to Close Valve</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Inlet Elevation</p> <p>C Junction Name</p> <p>S Loss Value</p> <p>S Outlet Elevation</p> <p>I Sizing/Cost Setting</p> <p>I Special Condition</p>	<p>S Design Factor</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Inlet Elevation</p> <p>C Junction Name</p> <p>S Loss When Fully Open</p> <p>S Outlet Elevation</p> <p>S Setpoint</p> <p>I Sizing/Cost Setting</p> <p>I Special Condition</p>	<p>Pressure</p> <p>S Initial Guess Temperature</p> <p>S Elevation</p> <p>C Junction Name</p> <p>I Sizing/Cost Setting</p>	<p>S Design Factor</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Inlet Elevation</p> <p>C Junction Name</p> <p>S K Factor</p> <p>S Outlet Elevation</p> <p>I Sizing/Cost Setting</p> <p>I Special Condition</p>
<p>Heat Exchanger</p> <p>S Controlled Outlet Temperature</p> <p>S Design Factor - Friction Loss</p> <p>S Design Factor - Heat Transfer</p> <p>S Heat Flow Into System</p> <p>S Heat Transfer Area</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Inlet Elevation</p>	<p>Jet Pump</p> <p>S Design Factor</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Inlet Elevation</p> <p>C Junction Name</p> <p>S Outlet Elevation</p> <p>I Sizing /Cost Setting</p> <p>I Special Condition</p>	<p>Orifice</p> <p>S Area/Diameter</p> <p>S Design Factor</p> <p>S Exit Pressure</p> <p>S Exit Temperature</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Inlet Elevation</p> <p>C Junction Name</p> <p>S Loss Value</p>	<p>Pump</p> <p>S Control Setpoint</p> <p>S Delta Pressure to Re-Open Combined Check Valve</p> <p>S Design Factor</p> <p>S Fixed Flow Rate</p> <p>S Fixed Speed (%)</p> <p>S Forward Velocity to Close Combined Check Valve</p>	<p>Relief Valve</p> <p>S Blowdown Pressure</p> <p>S Design Factor</p> <p>S Exit Head/Pressure</p> <p>S Exit Temperature</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Inlet Elevation</p> <p>C Junction Name</p> <p>S Loss Value</p>

Interface and User Options

<p>C Junction Name</p> <p>S K Factor</p> <p>S Outlet Elevation</p> <p>S Overall Heat Transfer Coefficient</p> <p>S Secondary Fluid Flow Rate</p> <p>S Secondary Fluid Inlet Temperature</p> <p>S Secondary Fluid Specific Heat</p> <p>I Sizing/Cost Setting</p> <p>I Special Condition</p>		<p>S Outlet Elevation</p> <p>I Sizing/Cost Setting</p>	<p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Inlet Elevation</p> <p>C Junction Name</p> <p>S Liquid Depth</p> <p>S Liquid Surface Pressure/HGL</p> <p>S Outlet Elevation</p> <p>I Sizing/Cost Setting</p> <p>I Special Condition</p> <p>S Submerged Suction Pressure</p> <p>S Submerged Suction Temperature</p> <p>S Trimmed Impeller Ratio as Percent</p>	<p>S Outlet Elevation</p> <p>S Overpressure</p> <p>S Remote Pressure</p> <p>S Set Pressure</p> <p>I Sizing/Cost Setting</p> <p>I Special Condition</p>
<p>Reservoir</p> <p>S Cross-Sectional Area</p> <p>S Design Factor</p> <p>C Junction Name</p> <p>S Liquid Surface Elevation</p> <p>S Liquid Surface Pressure</p> <p>S Liquid Tem-</p>	<p>Screen</p> <p>S Design Factor</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Inlet Elevation</p> <p>C Junction Name</p>	<p>Spray Discharge</p> <p>S Cd (Discharge Coefficient)</p> <p>S Design Factor</p> <p>S Discharge Flow Area</p> <p>S Elevation</p> <p>S Exit Pressure</p> <p>S Exit Tem-</p>	<p>Three Way Valve</p> <p>S Actual Percent Open</p> <p>S Design Factor</p> <p>S Elevation</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p>	<p>Tee or Wye</p> <p>S Design Factor</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Elevation</p> <p>C Junction Name</p> <p>I Sizing/Cost Setting</p>

Interface and User Options

<p>perature</p> <p>I Sizing/Cost Setting</p> <p>I Surface Elevation Type</p> <p>S Tank Bottom Elevation</p>	<p>S K Factor</p> <p>S Outlet Elevation</p> <p>I Sizing/Cost Setting</p>	<p>perature</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>C Junction Name</p> <p>I Number of Holes</p> <p>I Sizing/Cost Setting</p> <p>I Special Condition</p>	<p>C Junction Name</p> <p>I Sizing/Cost Setting</p>	
<p>Valve</p> <p>S Design Factor</p> <p>S Exit Pressure</p> <p>S Exit Temperature</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Inlet Elevation</p> <p>C Junction Name</p> <p>S Loss Value</p> <p>S Open Percentage</p> <p>S Outlet Elevation</p> <p>S Restricted Flow Area</p> <p>I Sizing/Cost Setting</p> <p>I Special Condition</p>	<p>Venturi</p> <p>S Design Factor</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Inlet Elevation</p> <p>C Junction Name</p> <p>S Loss Value</p> <p>S Outlet Elevation</p> <p>I Sizing/Cost Setting</p>	<p>Volume Balance</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Elevation</p> <p>C Junction Name</p> <p>I Sizing/Cost Setting</p>	<p>Weir</p> <p>S Channel Width</p> <p>S Downstream Bottom Elevation</p> <p>S Downstream Cross-Sectional Area</p> <p>S Height of Weir</p> <p>S Initial Guess Pressure</p> <p>S Initial Guess Temperature</p> <p>S Inlet Elevation</p> <p>C Junction Name</p> <p>S Notch Width/Angle</p> <p>S Outlet Elevation</p> <p>I Sizing/Cost Setting</p>	

			I Special Condition S Surface Pressure S Upstream Bottom Elevation S Upstream Cross-Sectional Area I Weir Type	
--	--	--	---	--

Change Code

Each Parameter has restrictions on how it can be changed. All of the Parameters are decimal numbers, integers, or strings.

Table 2: Valid Change Codes for each type of Parameter

Decimal Number (S)	Integer Number (I)	String (C)
Set equal to value	Set equal to value	Set equal to value
Change by value (+/-)	Change by value (+/-)	Add string to list
Change by percent (+/-)		Delete string from list

Value

The actual value to change the Parameter to or by.

For special conditions there are specific values assigned to each special condition option, as are listed in the table below.

Table 3: Values used to import special conditions

Junction	Special Condition	Value
Assigned Flow	None	0
	Closed	1
Branch	None	0
	Closed	1
Check Valve	None	0
	Closed	1
	Open	2
Control Valve	None	0
	Closed	1
	Fully Open - No Control	2
General Component	None	0
	Closed	1
Heat Exchanger	None	0
	Ignore Heat Transfer	2
Jet Pump	None	0
	Closed	1
Pump	None	0
	Pump Off No Flow	1
		2
	Pump Off With Flow Through	

Junction	Special Condition	Value
Relief Valve	None	0
	Failed	2
	Open	5
	Ignore Relief Valve	
Spray Discharge	None	0
	Closed	1
Valve	None	0
	Closed	1
Weir	None	0
	Ignore Weir	5

Scenario Path Name

The Scenario that the changes are intended to apply to. If no scenario is present, the changes will be applied to the current scenario.

It is important that the Scenario name is uniquely qualified - if there are multiple scenarios with the same name, the full Scenario Path Name can be used. Scenario Path Names can easily be copied from AFT Fathom by right-clicking the scenario name and selecting "Copy Scenario Path Name."

Importing the File

When the sheet has been completely filled out, it can be imported from File -> Import Excel Change Data. When importing the data, the Object Change Log will appear, listing the changes that were made and any errors that occurred during the import.

Table 4: Excel Importing Error Messages

Error	Description
Scenario name is not unique in the model (use full scenario path name)	There are multiple scenarios with this name. Use the fully qualified path name (e.g. Base Scenario\US Units\Pump A).
Invalid change type	The Change Code is not valid for the Parameter. For example, trying to change a string type with Change by Value (+/-).
Invalid parameter ID	The parameter given does not match the object type (e.g. Length for a junction).
Invalid parameter value	The value cannot be changed because the value is not valid. For example, a pump speed cannot be negative.
Object not found	A pipe or junction with the given ID was not found.
Parameter conflict	The value cannot be changed because it conflicts with another setting. For example, the loss value cannot be set when the junction is using a resistance curve.
Scenario not	No scenario by the

found in the model	given name was found in the model.
--------------------	------------------------------------

Note: In order for the junction to accept the changes, some fields must already contain data. For example, if the control setpoint for a control valve is being changed, there must already be a setpoint and units entered into the control valve junction object. This is to ensure that the correct set of units are being used. These values can be set using [Global Edit](#) to an arbitrary value before the changes are made from Excel.

Related Topics

File Import and Export	458
Importing Piping Component Files	458
Importing GIS Shapefiles	462
Importing CAESAR II Neutral Files	464
Importing and Exporting EPANET Files	467
Importing and Exporting from Other AFT Products	471
Exporting To Excel	472

Related Examples

NFPA Firewater System	934
-----------------------------	-----

Special Modeling Features

This section is dedicated to unique standard based outputs as well as the designation of design alerts. Topics include discussions on using the submergence parameter to prevent vortexing, and utilizing the design alerts feature in the Workspace.

Contents

Submergence	490
Design Alerts	491
Design Alerts From Output	494
Design Alerts Importing & Exporting	496

Submergence

Submergence is a parameter used to determine the amount of liquid needed above a pipe supply from a tank or reservoir to ensure smooth flow and avoid vortexing. AFT Fathom offers a pipe [output parameter](#) for submergence and also submergence margin, which is the difference between the submergence and the actual pipe depth.

Note: Pipe output parameters for submergence include options for Submergence Inlet and Submergence Outlet. Like all pipe output parameters, the Inlet and Outlet reference the inlet and outlet of the pipe. They do not describe if flow is entering or exiting the tank.

The equation used in AFT Fathom for submergence is given by [ANSI/HI Standard 9.8.7 1998](#), p. 32:

$$S = D(1 + 2.3 Fr)$$

Where S is the submergence and Fr is the Froude number. Note that the required submergence can be modified by using devices such as baffles in which case the above equation would not apply.

Related Topics

Design Alerts	491
Design Alerts From Output	494
Design Alerts Importing & Exporting	496

Design Alerts

Design Alerts are a powerful feature that can help speed up the modeling process and help verify that the model is behaving as expected. The creation of a Design Alert does not constrain or affect the operation of the model in any way. However, it automatically creates an output alert any time the design limit is violated. One common use case might be defining a maximum allowable pressure for the pipes in the model. While the output can be reviewed manually for these violations, this quickly becomes tedious as the model grows in size. Creating a Design Alert instead allows the engineer to spend less time reviewing output tables as any violation will be immediately identified.

The Design Alert Manager allows the user to create or modify Design Alerts. It can be accessed from the Tools Menu or in the Design Alerts tab in pipe and junction properties windows.

Design Alerts can also be [created directly from the output](#), or can be [imported/exported](#) for use in other models.

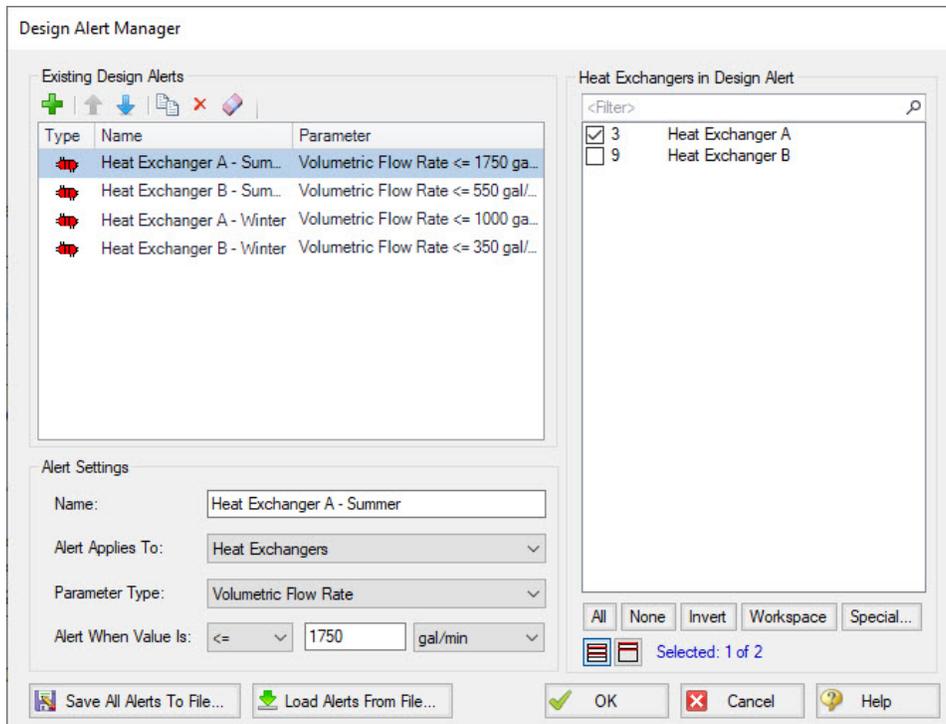


Figure 1: Design Alert Manager

Creating and Applying Design Alerts

Click the green plus sign (New Design Alert) to add a new Design Alert to the model.

A Name, Object Type, and Parameter Type and Value can be specified. Depending on the Object Type, a different selection of Parameters are available, as shown below. To add a Parameter:

Special Modeling Features

1. Give the Design Alert a distinct and descriptive name.
2. Select the Object Type the alert will be applied to.
3. Select the Parameter Type the alert will monitor.
4. Specify a value, unit if applicable, and companion.
 1. Use \geq when the value shouldn't rise above the specified value (i.e. temperature shouldn't exceed a maximum value)
 2. Use \leq when the value shouldn't fall below the specified value (i.e. pressure shouldn't drop below a minimum value)
5. Select which Objects from the Workspace the Design Alert will be applied to.
6. Click "OK" to save changes.

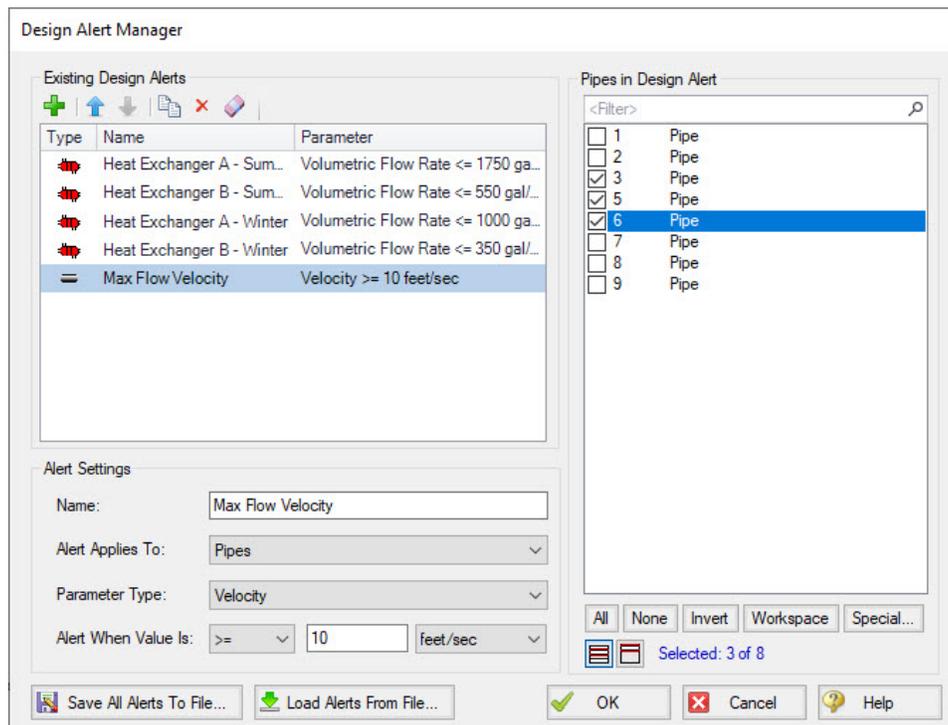


Figure 2: Creating a Design Alert

Table 1: Available Design Alerts

Pipes	General Junctions	Pump	Valves	Heat Exchangers	Reservoirs
Mass Flow Rate	Gradeline Energy Inlet (EGL)	Head Rise	Cv	Head Loss	Liquid Height
Pressure Gradient Stag. Total	Gradeline Energy Outlet (EGL)	Mass Flow Rate	Flow Area	Heat Rate	Liquid Surface Elevation
Pressure Stagnation	Gradeline Hydraulic Inlet (HGL)	NPSH Margin Absolute	Head Loss	Log Mean Temperature Difference	Mass of Liquid in Reservoir
Pressure Static	Gradeline Hydraulic Outlet (HGL)	NPSHA	K	Mass Flow Rate	Volume of Liquid in Reservoir
Submergence Inlet Margin	Pressure loss Stagnation Total	Percent of BEP	Kv	Pressure Drop Stagnation	
Submergence Outlet Margin	Pressure Loss Stagnation Total	Power Overall	Mass Flow Rate	Temperature Inlet	
Temperature Stagnation	Pressure Loss Static Total	Pressure Discharge Stagnation	Open Percentage	Temperature Outlet	
Vapor Pressure Margin	Pressure Stagnation Inlet	Pressure Discharge Static	Pressure Drop Stagnation	Volumetric Flow Rate	
Velocity	Pressure Stagnation Outlet	Pressure Rise Stagnation	Pressure Static Inlet		
Volumetric Flow Rate	Pressure Static Inlet	Pressure Suction Stagnation	Pressure Static Outlet		
	Pressure Static Outlet	Pressure Suction Static	Volumetric Flow Rate		
		Speed			
		Volumetric Flow Rate			

When running the model, the selected object's output values will be checked against the conditions that have been defined. If any of the Design Alert conditions are met, the user will receive a Design Alert warning on the Design Alerts tab in the general output section, and the violating parameters will be highlighted in the [Output](#) tables. Design Alert warnings can be double-clicked to take you to that object in the Workspace.

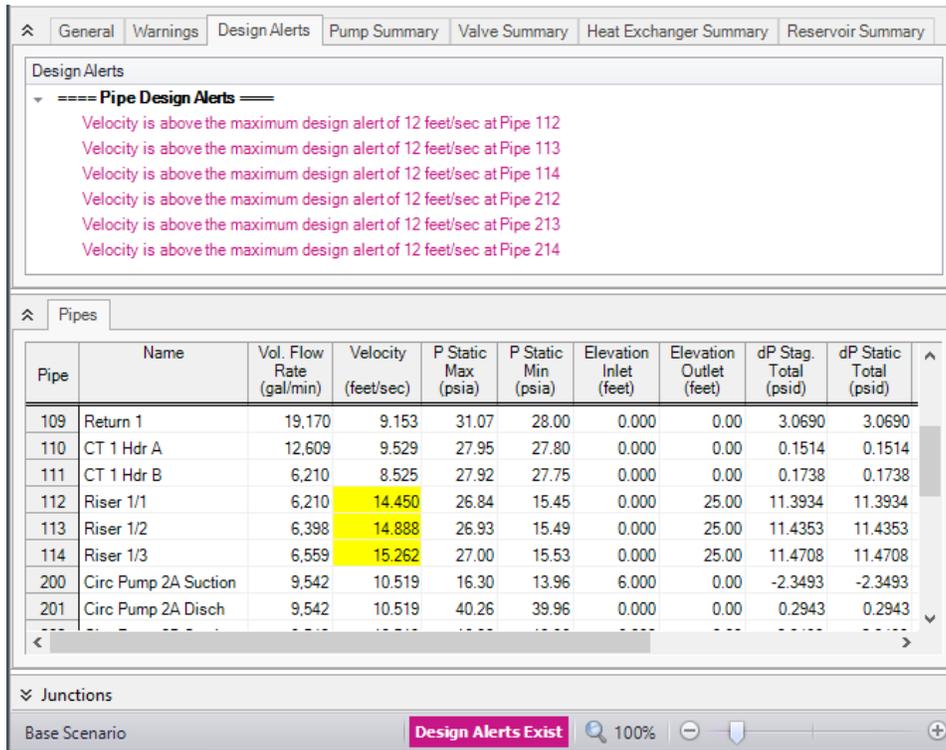


Figure 3: Design Alert Violations in Output

Design Alerts can be cross-plotted in the [Graph Results window](#) vs. the actual results. One good example where this is useful is for cross-plots of maximum and minimum allowed operating pressures. Design alerts are visible in every scenario in a model, but can be turned off or on for specific scenarios. When creating a design alert for the first time it will be active in that scenario and all descendent scenarios. For more information about scenarios, please read the [Scenario Manager](#) help topic.

Related Topics

Submergence.....	490
Design Alerts From Output.....	494
Design Alerts Importing & Exporting	496

Design Alerts From Output

Right-clicking on a cell in the Output Window brings up a context menu. One of these options is Create Design Alert...

Flow ID	Velocity (feet/sec)	P Static Max (psia)	P Static Min (psia)	Elevation Inlet (feet)	Elevation Outlet (feet)	dP Stag. Total (psid)	dP Static Total (psid)	dP Gravity (psid)
0.0	9.643	22.11	18.41	10.00	0	-3.7043	-3.7043	-4.338
5.0	4.821	102.54	102.29	0.00	0	0.2580	0.2580	0.000
5.3	7.264	73.75						0.00
5.0	4.821	21.50						0.00
1.7	7.249	101.17						0.00
5.5	9.665	72.76						0.00
8.3	6.533	100.47						0.00
0.2	6.555	75.31						0.00
5.0	4.821	22.58						0.00
0.0	9.643	101.49	100.91	0.00	0	0.5768	0.5768	0.000
5.0	4.821	102.47	102.21	0.00	0	0.2580	0.2580	0.000

Figure 1: Output Context Menu

Create Design Alert automatically defines a Design Alert based on the selected value and quantity:

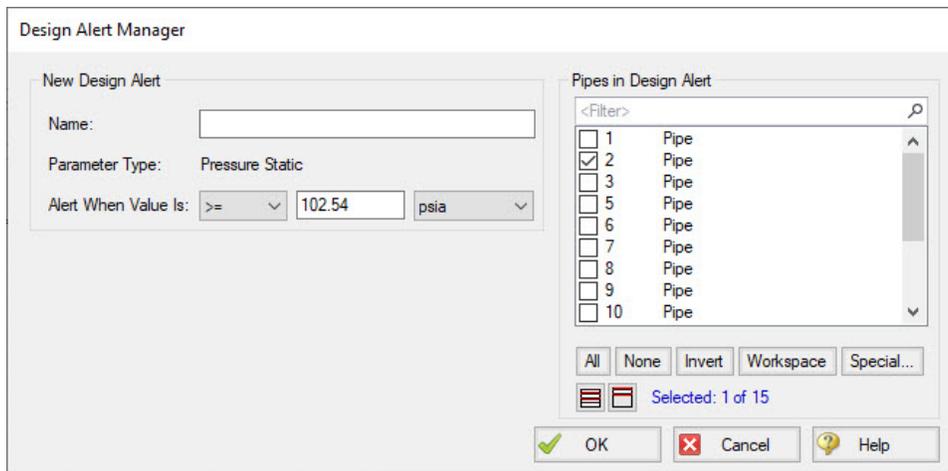


Figure 2: Automatically defined Design Alert

The Design Alert needs to have a name defined to be added. The specific value of the alert can be modified, and whether it is a maximum or minimum can be selected. The associated object is automatically selected as part of the Design Alert, but this is not required and additional objects can be selected now.

After the Design Alert has been created it can be further modified in the [Design Alert Manager](#).

Related Topics

Submergence.....	490
Design Alerts.....	491
Design Alerts Importing & Exporting.....	496

Design Alerts Importing & Exporting

Design Alerts can be saved to a text file to be imported into other model files. This allows the user to quickly load commonly used design alerts into multiple model files.

Export a Design Alert

To export a Design Alert:

1. Fully [define a Design Alert](#) for any parameter in the Design Alert Manager. For this example, we are using a velocity design alert.

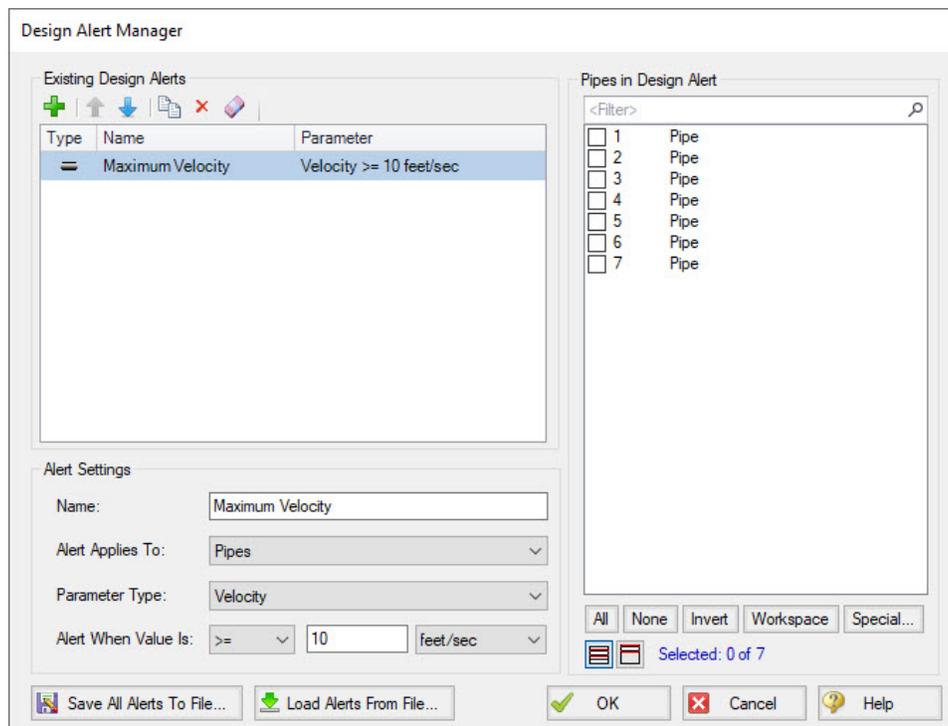


Figure 1: A design alert for velocity to be exported

2. Select "Save All Alerts To File..." and give the file a descriptive name.

Import a Design Alert

1. Open the Design Alert Manager from the Tools menu.
2. Select "Load Alerts From File...", then browse to and select the saved design alert file.
3. The Design Alert should now be added to the Existing Design Alerts list, and can be applied to objects in the model.

Related Topics

Submergence.....	490
Design Alerts.....	491
Design Alerts From Output.....	494

Scenario Manager

Scenario Manager (in the [Quick Access Panel](#)) is a powerful tool for managing variations of a model, referred to as scenarios. It allows you to quickly test the same model with selected differences (such as pump size, pipe diameter, system location, units, etc.) to find the system which works best. The Scenario Manager allows you to:

- Create, name and organize scenarios
- Select the scenario to appear on the [Workspace](#) (the "current" scenario)
- Delete, copy, and rename scenarios
- Duplicate scenarios and save them as separate models
- Review the source of a scenario's specifications
- Pass changes from a scenario to its variants
- See what scenarios have output by automatically changing the text color to blue if they have output

All scenarios are saved within the model file.

Scenario

A variant case of a model that is created in the Scenario Manager.

Base Scenario

The Base Scenario is the root scenario at the top of the scenario tree. As the highest level ancestor, It does not inherit from any other scenario. A new scenario can be inserted above the base scenario, at which point the new scenario would behave as the base scenario, and the original base scenario would behave as a child scenario. "Base Scenario" is the default name given in a new, single scenario model.

The Scenario Manager

Illustrated in Figure 1, the top of the Scenario Manager displays the scenario tree, showing the names of the currently defined scenarios, their relationship to other scenarios, and which is the current Workspace scenario (the current scenario is identified both as the scenario with a green check mark next to it and in AFT Fathom's [Status Bar](#)). At the bottom, there is a place where you can record notes related to that scenario (propagates to children).

Those attributes whose value comes from a different scenario are linked, while those values coming from the selected scenario are not linked. Linked attributes will inherit changes made in the same object and attribute of the linked ancestor. This inheritance will occur over as many generations as the link exists.

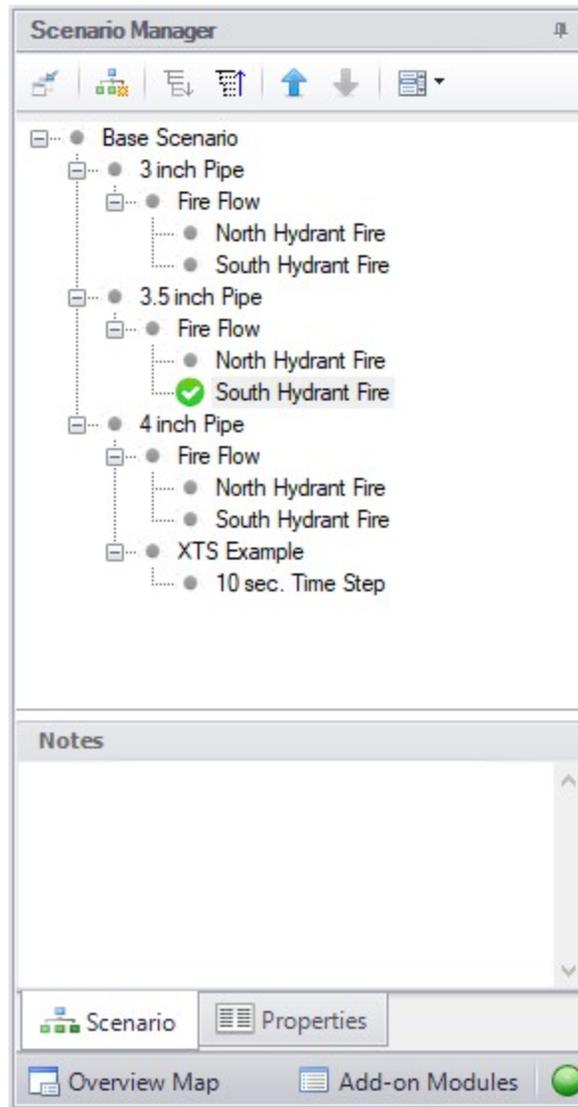


Figure 1: Scenario Manager on Quick Access Panel

Related Topics

Note: No scenarios have been set in the Multi-Scenario Output display..... 788

Creating, Organizing, and Editing Scenarios..... 500

Modifying Individual Scenarios..... 503

Scenario Inheritance..... 504

Running Models in Batch Mode..... 508

Open Model Using Last Scenario..... 509

Disconnected Scenario..... 510

Scenario Comparison Tool Overview..... 510

Related Examples

Housing Project 900

Creating, Organizing, and Editing Scenarios

Scenarios are referred to as either a parent or child, plus the [Base Scenario](#), which may be thought of as the root scenario. Upon opening a new model file, only a Base Scenario will be present. To create a new scenario below the Base Scenario, right-click the Base Scenario and select Create Child.

Any scenario can be made current by double-clicking on it.

Figure 1 illustrates a model with a Base Scenario with two children, "English Units" and "Metric Units". These have three children, '3 inch Pipe', '3.5 inch Pipe' and '4 inch Pipe', representing three different sized pipe runs. Each of these has one child scenario "Fire Flow" and each of these, in turn, has two children, 'North Hydrant Fire' and 'South Hydrant Fire', for the two operating cases.

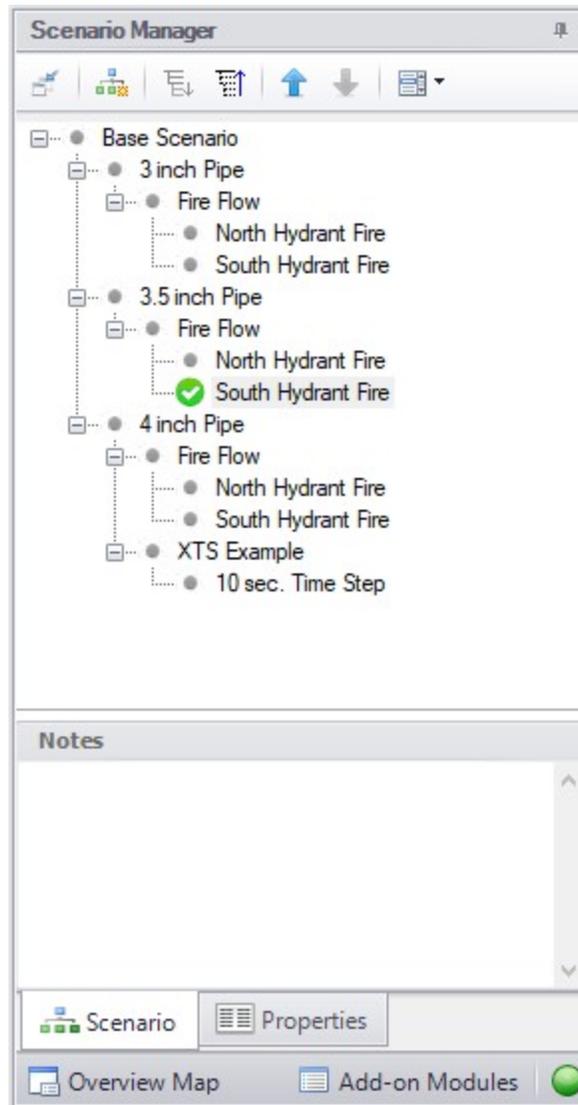


Figure 1: Scenario Manager Window

'English Units' and 'Metric Units' were created by first selecting the Base Scenario and Creating a Child. '3 inch Pipe', '3.5 inch Pipe' and '4 inch Pipe' were created by first selecting the 'English Units' Scenario then Creating Children. 'Fire Flow' was created by clicking on '3 inch Pipe' and creating the child. 'North hydrant open' and 'South hydrant open' were, in turn, created by selecting each of the above children scenarios for the three different pipe run sizes, then Creating Children.

Note: Alternatively, one could have made the 'English Units' scenario current, created the '3 inch Pipe' child, then selected it and created the 'North Hydrant Fire' and 'South Hydrant Fire' scenarios below it, then, with '3 inch Pipe' still selected, selected Clone With Children twice, then rename the new clones '3.5 inch Pipe' and '4 inch Pipe'.

The 'North Hydrant Fire' and 'South Hydrant' scenarios may be thought of as great great grand children of the Base Scenario. This concept of scenario ancestry is helpful in keeping track of the differences and similarities between scenarios discussed later.

A box with a plus sign next to a scenario indicates it may be expanded to reveal children scenarios while a box with a minus sign indicates the level is fully expanded with all children scenarios shown, if any. No box indicates there are no children. Levels may be expanded by clicking on the plus sign and condensed by clicking on the minus sign.

Right-clicking a scenario provides the following choices:

- **Create Child with No Output** - create child scenarios without results saving duplication and load time
- **Rename** - renames the selected scenario
- **Delete Scenario** - deletes the selected scenario
- **Delete All Children** - deletes all children of the selected scenario
- **Clone Without Children** - creates a copy of the selected scenario at the same level (a sibling). Cloning a scenario prompts the user for a name for the new scenario
- **Clone With Children** - creates a copy of the selected along with its children. Cloning a scenario prompts the user for a name for the new scenario.
- **Insert Scenario Above Selected...** - creates a new scenario where the selected scenario is and shifts the selected scenario and its children down a level. This can be used to create a new base scenario. If this is done the user will be prompted to rename the previous base scenario.
- **Promote** - raises a scenario one level, i.e. from a child to a sibling, active only for scenarios more than one level below the base scenario
 - **Copy/Save Scenario Names**
 - **Copy Scenario Path Name** - copy the fully-qualified name of the scenario to the Windows clipboard ("Base Scenario/Case 1/Child 3")
 - **Copy Scenario Path Name With File Name** - copy the fully-qualified scenario name along with the model name ("Model.fth :: Base Scenario/Case 1/Child 3")
 - **Save Scenario Names Only to File...** - copy all scenario names on separate lines, with indents to show child/parent relations
 - **Save Full Scenario Paths to File...** - copy all scenario names on separate lines, with the full path name used for each scenario
- **Save Scenario to File Without Children** - saves the selected scenario to a separate file without its children
- **Save Scenario to File With Children** - saves the selected scenario and its children to a separate file
- **Compare to Active Scenario** - compares all parameters for all objects between the selected scenario and active scenario and displays the [Comparison Grid](#) showing the results
- **Scenario Comparison Tool** - opens the [Scenario Comparison tool](#) to create a custom scenario comparison

Note: It is important to remember that AFT Fathom’s Edit menu Undo does not apply to Scenario Manager operations. To undo a Rename scenario operation, you can rename the scenario again to the old name. To undo a Clone or Save Scenario operation, you can delete the clone or file created. Once a scenario is deleted or promoted, there is no way to un-delete or un-promote the scenario.

Viewing Scenario Differences

Differences between scenarios can be viewed in two different ways. All data associated with any selection of scenarios can be viewed using the [Scenario Comparison tool](#). Alternatively, a scenario’s data compared to all of its direct ancestors can be viewed in the [Model Data window](#). This functionality can be enabled in the [Model Data Control](#) window.

Output from different scenarios can also be displayed in the Output window using [Output Control](#) features.

Fast Scenario Changes

AFT Fathom remembers the most recent scenario you had open, and allows you to quickly change back to that scenario by using the Last Scenario feature. The Last Scenario feature is found on the [Tools menu](#) and the [Common Toolbar](#).

Related Topics

Scenario Manager.....	498
Modifying Individual Scenarios.....	503
Scenario Inheritance.....	504
Running Models in Batch Mode.....	508
Open Model Using Last Scenario.....	509
Disconnected Scenario.....	510
Scenario Comparison Tool Overview.....	510

Modifying Individual Scenarios

When first created, a scenario is identical to its parent. To modify or run a scenario, it must first be made in the current Workspace scenario by selecting it within the Scenario Manager window, then clicking on the Load As Current Scenario button. (The name of the current scenario is displayed in both the Scenario Manager window and in AFT Fathom’s [Status Bar](#).) Once a scenario is the current scenario, changes may be made to differentiate it from its parent using any of AFT Fathom’s editing tools and functions. When an attribute is changed within a scenario, it breaks the link for this attribute. Subsequent changes in the scenario’s ancestors will no longer affect this attribute.

Properties of a scenario that may differ from its ancestors include:

- Presence and location of objects -- one may add or delete pipes and junctions within a scenario and the workspace locations may vary from scenario to scenario
- [Pipe](#) and [junction](#) Properties - All values set in the Properties windows
- [Fluid Properties](#) - Fluid, viscosity model, heat transfer, and laminar and non-Newtonian corrections
- [Steady Solution Control](#) - Solution method, tolerances, relaxation
- [Output Control](#) - Output window content and formatting
- [Cost Settings](#) - Application of cost data
- [Transient Control \(XTS Module Only\)](#) - Start and stop times for transient simulation
- [Goal Seek & Control \(GSC Module Only\)](#) - Application of variables and goals can change between scenarios. The actual variables and goals are the same in all scenarios.
- [Library Manager](#) - The connected library content can be varied between scenarios

[Model Data Control](#) and [User Options](#) apply to all scenarios.

Related Topics

Scenario Manager.....	498
Creating, Organizing, and Editing Scenarios.....	500
Scenario Inheritance.....	504
Running Models in Batch Mode.....	508
Open Model Using Last Scenario.....	509
Disconnected Scenario.....	510
Scenario Comparison Tool Overview.....	510

Scenario Inheritance

Scenario Manager will pass changes made in a scenario to linked attributes of its descendants, allowing multiple scenarios to be modified in one operation and maintain similarity between scenarios where desired.

Any attribute whose value is the same as that of the scenario's parent is linked to the parent. In turn, if the parent's attribute value is the same as that of its parent, then the child's attribute is linked to its grandparent. This ancestral linking of attributes can exist across any number of generations. A change made to a scenario will be passed downward to its children, grandchildren, etc., as far as the link exists. Changes are NOT passed upward from child to parent.

Once an attribute is changed within a scenario, the link is broken to that individual attribute. Subsequent changes in the scenario's ancestors, parent, grandparent, etc., will no longer affect that attribute, but all other attributes within the scenario will remain linked. The [Model Data window](#) with Scenarios enabled shows how the data changes between scenarios.

Scenario Logic Examples

For many users, it is easiest to grasp Scenario Manager when it is explained how the coding logic is actually implemented. **Blank fields for children, grandchildren, etc., mean to look to the parent for the data.** The Base Scenario never has blank fields (Table 1). Data only passes downwards, never upwards.

Table 1: Blank fields for Child #1 and Grandchild #1 mean that the data is to come from the parent. If the Base scenario data is changed, all descendants are changed.

	Diameter	Length
Base	3	25
Child #1	--	--
Grandchild #1	--	--

If a child scenario does not have a blank field, then data for that property is initiated at that scenario level (see Table 2).

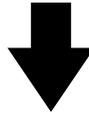
Table 2: Child #1 does not have a blank field, so it's Diameter would be 2, not 3, as would Grandchild #1

	Diameter	Length
Base	3	25
Child #1	2	--
Grandchild #1	--	--

If a scenario is changed, and its child has different data, then the change will not pass downwards (see Table 3).

Table 3: Changing the Base Scenario Diameter from 3 to 6 would not impact Child #1 or any descendants in that line. Changing the Length from 25 to 40 would also change the length in Child #1, Grandchild #1, and any descendants of Grandchild #1.

	Diameter	Length
Base	3	25
Child #1	2	--
Grandchild #1	--	--



	Diameter	Length
Base	6	40
Child #1	2	--
Grandchild #1	--	--

If a child scenario has data that is different than the parent, its children cannot relink to the parent (see Table 4).

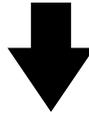
Table 4: Even if the Grandchild #1 has the same Diameter as the Base, it is not linked to the Base because it and its parent are not blank.

	Diameter	Length
Base	3	25
Child #1	2	--
Grandchild #1	3	--

If a child scenario's data, which was previously changed and is thus different from the parent, is changed back to the same value as the parent, the inheritance link is re-established (see Table 5). Its descendant's link is also re-established.

Table 5: If the Diameter in Child #1 is changed to be the same as the Base, it will be "blanked out" the next time the scenario is loaded and the link re-established. And so will Grandchild #1, if its Diameter is also the same

	Diameter	Length
Base	3	25
Child #1	3	--
Grandchild #1	3	--



	Diameter	Length
Base	3	25
Child #1	--	--
Grandchild #1	--	--

Table 6 shows data for two pipe properties across three scenarios. The data that would be used when each scenario is loaded is as follows:

- Base scenario
 - Diameter = 3
 - Length = 25
 - Changes to Diameter will not pass downwards
 - Changes to Length will affect only Child #1.
- Child #1 scenario
 - Diameter = 2
 - Length = 25
 - Changes to Base Diameter will not affect Diameter
 - Changes to Base Length will affect Length
- Grandchild #1 scenario
 - Diameter = 2
 - Length = 15
 - Changes to Base Diameter will not affect Diameter
 - Changes to Child #1 Diameter will affect Diameter
 - Changes to Base Length or Child #1 Length will not affect Length

Table 6: Input data for these properties explained in the text.

	Diameter	Length
Base	3	25
Child #1	2	--
Grandchild #1	--	15

Re-Establishing Broken Scenario Links

A link may be re-established by returning the attribute to the same value as that of its parent. This can be done manually by entering the value or selecting the Copy Data From Pipe list and selecting the Parent Pipe Data option. Within other data windows, it is done by selecting the Same As Parent option in Output Control, Analysis Setup, or Library Manager

Since links are identified by comparing attribute values of pipes or junctions with the same Workspace ID number, renumbering a scenario will break the links of all pipes and junctions renumbered. Since numbers must be unique, once a link has been broken by renumbering, it may not be re-established.

Related Topics

Scenario Manager.....	498
Creating, Organizing, and Editing Scenarios.....	500
Modifying Individual Scenarios.....	503
Running Models in Batch Mode.....	508
Open Model Using Last Scenario.....	509
Disconnected Scenario.....	510
Scenario Comparison Tool Overview.....	510

Running Models in Batch Mode

AFT Fathom [scenarios](#) or models can be run in batch mode, a feature that is especially useful for running a group of scenarios or models during lunch or overnight.

To run models in batch mode, follow these steps:

1. From the File menu, click Start Batch Run.
2. Select the Batch Run Type. This is where you select if you want to run scenarios from the current model or different model files. If the current model does not have any scenarios, then only the second option will be available.
3. If the Batch Run Type is Scenarios in Current Model, then click the Add Scenarios button and select the scenarios you wish to run.
4. If the Batch Run Type is Models from Different Files, then the models can be selected individually using the Add Model Files button, or loaded from a Batch File (by Choosing the Load Existing

Batch File button). A Batch File is a text file listing all models you would like to run. At any time, a selected model file can be removed by clicking the Remove Selections button. The current list of models can be saved to a Batch File for use in the future by clicking the Save List to File button.

5. The option can be chosen to Save Using Excel Export Manager by either saving to separate sheets, or saving the scenarios to separate workbooks. The Excel Export option can only be chosen if the batch run type is for Scenarios in Current Model. Additionally, the desired output must first be configured in the [Excel Export Manager](#) under the File menu for any data to be exported.
6. Use the Output Options in the lower left to save the output data to a file and/or send it to a printer or Adobe PDF file. All reports will use the specified font.
7. Check the box to Run Batch in Background if you wish for all selected scenarios to run without showing the Solution Progress window. AFT Fathom will run the simulations in the background and will not interrupt any work being done in other programs.
8. Click the Start Run button to begin the batch run.

AFT Fathom then opens each scenario or model file in sequence, automatically runs each scenario/model that has [Analysis Setup](#) fully defined, and sends results to the specified destination. All pertinent information that is generated during the batch run (such as error messages) are automatically displayed when the final scenario/model is finished. Output files generated from each scenario or model run is automatically saved before moving to the next run.

The entire batch run can be canceled at any time by clicking the Cancel button in the Solution Progress window. Output from scenarios that have already been run is still retained when the Batch Run is canceled.

Note: You can view output from multiple scenarios concurrently in the [Output window](#).

Related Topics

Scenario Manager.....	498
Creating, Organizing, and Editing Scenarios.....	500
Modifying Individual Scenarios.....	503
Scenario Inheritance.....	504
Open Model Using Last Scenario.....	509
Disconnected Scenario.....	510
Scenario Comparison Tool Overview.....	510

Open Model Using Last Scenario

When the model is saved, the current [scenario](#) is remembered, and by default, reloaded when the model is opened again. Your preference of whether to always open the model to the most recent scenario or to the base scenario can be specified in the [User Options](#) window.

Related Topics

Scenario Manager.....	498
Creating, Organizing, and Editing Scenarios.....	500

Modifying Individual Scenarios.....	503
Scenario Inheritance.....	504
Running Models in Batch Mode.....	508
Disconnected Scenario.....	510
Scenario Comparison Tool Overview.....	510

Disconnected Scenario

A special type of scenario that is no longer connected to the [Scenario Manager](#), created from the [Tools Menu](#) or from the Toolbar in the Output window. A disconnected scenario itself breaks the inheritance of normal child scenarios so that it is truly standalone scenario. *No* changes made to any parent scenario will propagate to this scenario. This is useful to preserve certain settings from a different scenario.

Disconnected Scenarios are denoted with (*).

Related Topics

Scenario Manager.....	498
Creating, Organizing, and Editing Scenarios.....	500
Modifying Individual Scenarios.....	503
Scenario Inheritance.....	504
Running Models in Batch Mode.....	508
Open Model Using Last Scenario.....	509
Scenario Comparison Tool Overview.....	510

Scenario Comparison Tool OverviewScenario Manager

Scenario Comparison Tool (accessed via the Tools menu) is a powerful tool that allows users to compare model parameters between scenarios in a given model. It lets users quickly identify differences between two or more scenarios without needing to search through the model data for each scenario or inspect each element of the model. This tool can be especially powerful for users reviewing others' models, or for users returning to models after a period of time. It can also be powerful for troubleshooting purposes. The Scenario Comparison Tool lets you:

- Compare any two or more scenarios
- Compare selected Pipes and Junctions between selected scenarios
- Compare selected Pipe and Junction parameters between selected scenarios
- Compare selected General Properties between selected scenarios

Related Topics

Accessing Scenario Comparison Tool.....	511
Modify Comparison.....	512
Scenario Comparison Grid.....	513
Scenario Manager.....	498

Creating, Organizing, and Editing Scenarios.....	500
Modifying Individual Scenarios.....	503
Scenario Inheritance.....	504
Running Models in Batch Mode.....	508
Open Model Using Last Scenario.....	509
Disconnected Scenario.....	510

Accessing Scenario Comparison Tool

The Scenario Comparison tool can be accessed from the Scenario Manager, the Workspace, and the Tools menu.

From the Scenario Manager

To set up a comparison from the Scenario Manager, first [select](#) the pipes and junctions to compare in the workspace. Then, select the desired scenario to compare the current scenario to in the Scenario Manager. Note the current scenario is indicated by the green check-mark symbol. Right-click the desired scenario, and select Compare Selected Scenarios.... This will bring users to the Scenario Comparison window and display a [grid](#) with the differences between the selected objects in the two scenarios.

To modify the comparison use the [Modify Comparison](#) button at the bottom of the window.

From the Workspace

From the workspace, users can quickly select desired pipes and junctions in the scenario and compare them to those found in any other scenario.

First, select the desired pipe(s) and/or junction(s) in the workspace. Then, right-click the Workspace, choose Compare Scenarios, and select the desired comparison. This will bring users to the Scenario Comparison window and display a [grid](#) with the differences between the selected objects in the two scenarios.

To modify the comparison use the Modify Comparison button at the bottom of the window.

From the Tools Menu

From the [Tools Menu](#), there are multiple options to generate the Scenario Comparison Grid.

A comparison grid can be directly generated by selecting the desired objects to compare in the Workspace, then selecting the scenarios to be compared from the Compare Scenarios menu option.

Alternatively, the Scenario Comparison Tool menu option can be selected to build a custom scenario comparison by directly selecting the scenarios, objects, and parameters to be included in the comparison. See [Modify Comparison](#) for more information on the available options.

Click the Show Comparison button at the bottom of the window to view the comparison. To make additional changes, use the Modify Comparison button at the bottom of the window.

Related Topics

Scenario Comparison Tool Overview.....	510
Modify Comparison.....	512
Scenario Comparison Grid.....	513

Modify Comparison

Several Options are available to customize the [Scenario Comparison Grid](#). These options are available from the Modify Comparison button at the bottom of the scenario comparison report, or by selecting Scenario Comparison Tool from the [Tools menu](#). The Modify Comparison window is shown in Figure 1 below.

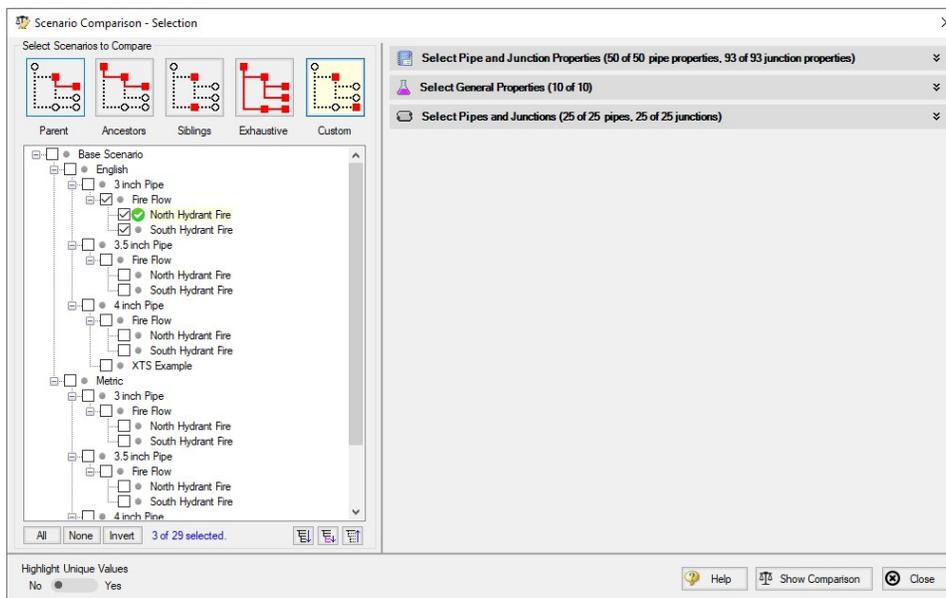


Figure 1: Scenario Comparison window options to Modify Comparison

Select Scenarios to Compare

The Scenario Comparison Tool has four pre-set templates for comparing scenarios, plus the ability to set up a custom comparison. These are:

- **Parent:** This template compares the current scenario with that scenario's parent.
- **Ancestors:** This template compares the current scenario with all of that scenario's direct ancestors. This setting would be similar to the comparison available in the [Model Data window](#), though it does provide a more comprehensive comparison.
- **Siblings:** This template compares the current scenario with all of that scenario's siblings.
- **Exhaustive:** This template compares all of the scenarios in the model with each other.
- **Custom:** This option allows users to compare any two or more scenarios throughout the model.

Select Pipe and Junction Properties

This section sets what parameters will be compared such as length, diameter, roughness, etc.

Users can choose to compare a set of specified properties, or all pipe and junction properties by selecting the desired properties under Select Pipe and Junction Properties. Only properties selected in this section will be considered when the comparison is run.

A helpful option to filter the parameters to be displayed in the comparison is to display the parameters shown in the Model Data window. This can simplify the comparison, but might hide some differences between scenarios. Use the [Model Data Control](#) window to further define what parameters are shown with this option.

Select General Properties

This section sets what General Properties will be compared such as fluid properties, tolerances, etc.

Users can choose to compare all or a select set of general properties from the [Analysis Setup window](#).

Select Pipes and Junctions

This section sets which pipes and junctions will be compared for the model.

Users can choose to compare all objects in the model, or a selected set of pipes and junctions. Users can manually select the desired objects, select those highlighted in the workspace, or use the [Select Special](#) tool to select desired groups or other sets of objects.

Related Topics

Scenario Comparison Tool Overview.....	510
Accessing Scenario Comparison Tool.....	511
Scenario Comparison Grid.....	513

Scenario Comparison Grid

Choosing the Compare Selected Scenarios menu item from the right-click or Tools menu will bring the user directly to the Scenario Comparison Grid based on any selected Workspace objects and scenarios. The settings for the scenario comparison can be seen/modified by clicking [Modify Comparison](#) at the bottom of the window.

The tabular data from the report can be copied from and pasted to a separate file or worksheet using the Copy button at the bottom of the window.

Scenario Manager

Item	Parameter	Scenario		
		North Hydrant Fire	3 inch Pipe	3.5 inch Pipe
P209		No Differences		
P210		No Differences		
P299	Hydraulic Diameter	3.068 inches	3.068 inches	3.548 inches
	Scaled Hydraulic Diameter	3.068 inches	3.068 inches	3.548 inches
	Size	3 inch	3 inch	3-1/2 inch
Junctions (25 of 25 compared)				
J1 (West Lake)		No Differences		
J101 (Branch)		No Differences		
J102 (Branch)		No Differences		
J103 (Branch)		No Differences		
J104 (Branch)		No Differences		
J105 (Branch)		No Differences		
J106 (Branch)		No Differences		
J107 (Browns)	Flow	2 gal/min	10 gal/min	10 gal/min
J108 (Peters)	Flow	2 gal/min	10 gal/min	10 gal/min
J109 (Chambers)	Flow	2 gal/min	10 gal/min	10 gal/min
J110 (Fords)	Flow	2 gal/min	10 gal/min	10 gal/min
J111 (Kellys)	Flow	2 gal/min	10 gal/min	10 gal/min
J199 (North Hydrant)	Special Condition	None	Flow Off	Flow Off
J201 (Branch)		No Differences		

Figure 1: Default Scenario Comparison Report view

Filtering Options

When a large comparison is run, typically many of the objects/parameters in the model will be unchanged across scenarios, which may result in a large number of items with no differences. To help condense the Scenario Comparison Grid, several toggles are available at the top of the Item and Parameter columns which allow the user to show/hide rows in the grid. Switching the toggle to All will show all items or parameters being compared. If the toggle is set to differences, then only Items/Parameters that have a different value in at least one scenario will be shown. Thus, if both toggles are set to All, then all parameters will be displayed for all items. If both toggles are set to Differences, then only items that have parameters with different values will be shown, and only the parameters that differ will be displayed.

Color Coding

The cells in the Item and Parameter columns will be colored either yellow or green to indicate whether that item/parameter has differences between the scenarios being compared. Yellow indicates differences, while green indicates that no differences exist. Parameters will be colored yellow if the value of that parameter is different in at least one scenario that is being compared. Items will be colored yellow if at least one parameter that is being compared for that item has a different value in one or more scenarios.

The Highlight Unique Values toggle is used to color the values to visually link identical values. All values under the currently active scenario will remain grey, while values that have differences will be colored. This is powerful to quickly see differences when more than two scenarios are being compared. If two scenarios have the same value for a parameter, the values will have the same shading. See Figure 2 below for an example.

Scenario Manager

Item Show		Parameter Show		Scenario		
Differences <input checked="" type="checkbox"/> All		Differences <input checked="" type="checkbox"/> All		North Hydrant Fire	3 inch Pipe	3.5 inch Pipe
P209				No Differences		
P210				No Differences		
P299		Hydraulic Diameter	3.068 inches	3.068 inches	3.548 inches	
		Scaled Hydraulic Diameter	3.068 inches	3.068 inches	3.548 inches	
		Size	3 inch	3 inch	3-1/2 inch	
<input checked="" type="checkbox"/> Junctions (25 of 25 compared)						
J1 (West Lake)				No Differences		
J101 (Branch)				No Differences		
J102 (Branch)				No Differences		
J103 (Branch)				No Differences		
J104 (Branch)				No Differences		
J105 (Branch)				No Differences		
J106 (Branch)				No Differences		
J107 (Browns)		Flow	2 gal/min	10 gal/min	10 gal/min	
J108 (Peters)		Flow	2 gal/min	10 gal/min	10 gal/min	
J109 (Chambers)		Flow	2 gal/min	10 gal/min	10 gal/min	
J110 (Fords)		Flow	2 gal/min	10 gal/min	10 gal/min	
J111 (Kellys)		Flow	2 gal/min	10 gal/min	10 gal/min	
J199 (North Hydrant)		Special Condition	None	Flow Off	Flow Off	
J201 (Branch)				No Differences		
J202 (Branch)				No Differences		

Only parameters with differences will be shown. If all chosen parameters for an item are the same, then the item will say 'No Differences'. Parameters containing tabular data (e.g. polynomial constants) can be viewed by right-clicking the parameter's row.

Item	Parameter	Highlight Unique Values
<input checked="" type="checkbox"/> All Parameters Same Value	<input checked="" type="checkbox"/> Same Value Across Scenarios	No <input checked="" type="checkbox"/> Yes
<input checked="" type="checkbox"/> At Least 1 Parameter Difference	<input checked="" type="checkbox"/> Different Values Across Scenarios	

Copy Help Modify Comparison Close

Figure 2: Scenario comparison showing differences only. Each unique value is highlighted with a different color to indicate which scenarios have differences from the currently active scenario, which is the North Hydrant scenario for this example.

Related Topics

Scenario Comparison Tool Overview.....	510
Accessing Scenario Comparison Tool.....	511
Modify Comparison.....	512

Cost Calculation

AFT Fathom has the powerful ability to account for both recurring and non-recurring costs in your model.

The most common and easiest to account for recurring cost is [Energy Cost](#). It is often desirable to reduce the electricity usage of pumps in a system, and AFT Fathom makes measuring this value easy.

In addition to energy usage, AFT Fathom can track other user-defined costs such as maintenance, installation, and material cost with a [Cost Library](#).

Cost Calculation Procedure

Depending on what types of costs or what components of a model you wish to include this process may change.

Note: This process only applies if the [ANS Module](#) is not enabled. To set up cost calculations with ANS, see the [Assign Cost Libraries](#) topic.

1. [Enable Cost Calculation](#) from the [Cost Settings panel](#) in Analysis Setup.
2. Specify the system life and energy cost in the [Cost Settings panel](#).
3. Junctions or pipes you wish to include in the cost calculation must be added to the Cost Report with the Cost tab in the object's Properties window.
4. If you are calculating all costs (not just Energy), you will need to create [Cost Libraries](#). To represent more complex Energy Cost, such as varying cost over time, you will need to create an [Energy Cost Library](#).
5. Within the Cost Libraries, specific recurring or non-recurring costs will need be [defined](#).
6. These costs can be further refined to adjust automatically based on diameter, time, power, or C_v with [Scale Tables](#).
7. Review the application of costs with the [Cost Application Manager](#) and the [Cost Summary window](#). It is important that the costs are being applied correctly.
8. Run the model and view the cost totals with the [Cost Report](#).

Related Topics

Cost Settings Group.....	170
Cost Settings Panel.....	171
Cost Application Manager.....	521
Scale Tables.....	523
Select Size From.....	525
Cost Definitions.....	526
Cost Libraries.....	529
Cost Summary Window.....	533
Custom Monetary Units.....	534
Cost Report.....	535

Related Examples

Beginner - Cost Calculation	908
Plant Cooling Cost Calculation	958

Enabling Cost Calculation

Note: This topic does not apply for the [ANS module](#). To set up cost calculations with ANS, see the [Assign Cost Libraries](#) topic.

The first step in any cost calculation is to enable it in the [Cost Settings panel](#). The following options are available:

- **Do not calculate** - Selected by default.
- **Pump Energy Only** - Only calculate the costs associated with the power consumption of pumps. The energy cost and system life must be entered in [Cost Settings](#).
- **Calculate** - Enable full calculation of costs. In addition to defining Cost Settings, [Cost Libraries](#) must be defined to use this feature.

Including Objects in Cost Report

Workspace objects must be included in the Cost Report to have their costs calculated. This allows only analyzing the cost of certain parts of the model - useful if only one pump is being replaced, or if an expansion to an existing system is being considered. To include the objects in the Cost Report simply select "Include Cost in Report" on the object's Cost tab, as shown below. Finer detail in how the Cost is included can be set by clicking the [Cost Application](#) button.

Cost Calculation

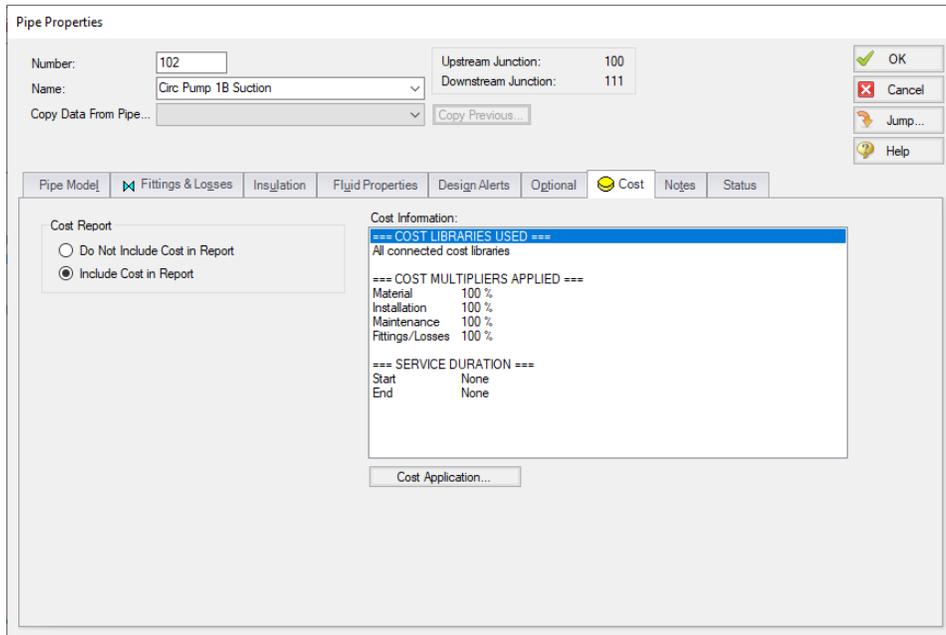


Figure 1: Include Cost in Report

Energy Cost Library

Energy Cost Libraries allow users to input time varying energy costs. These energy costs can then be assigned to [pumps](#), so that pump operation/energy costs can be calculated.

Creating an Energy Cost Library

When opening the Energy Cost Library window from the Library menu, you will be greeted with Figure 1. An Energy Cost Library either needs to be created or opened.

Cost Calculation

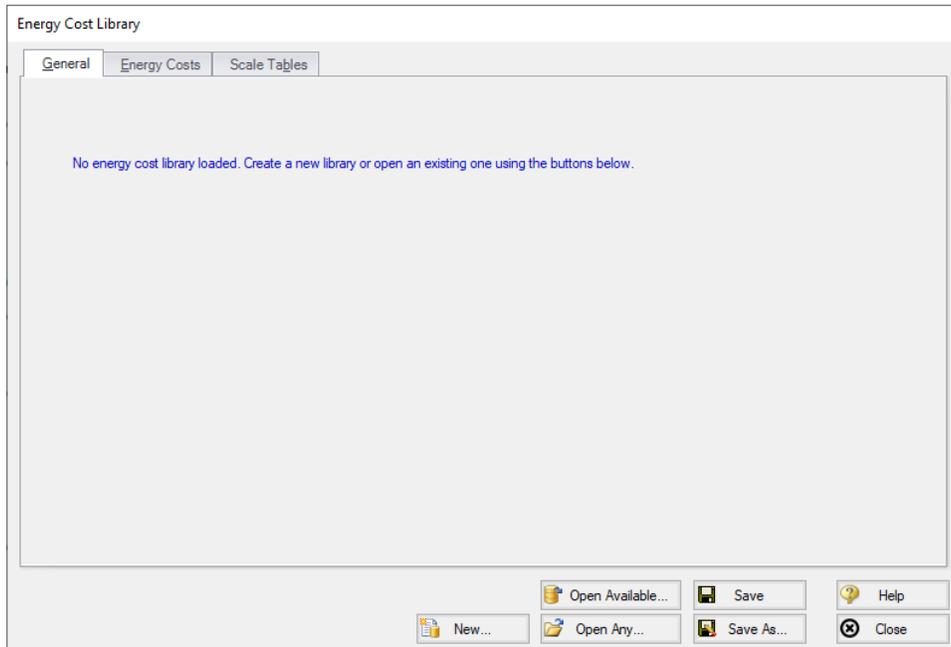


Figure 1: Energy Cost Library window

To create a new Energy Cost Library, click the New button. The Description field is mandatory, and the notes are optional. Select an alternate Monetary Unit if desired. Finally, save the library by clicking Save As and enter name for the *.ecd file. AFT Fathom will also prompt if the library should be connected and available - typically this is desired so the library can be used in the currently open model.

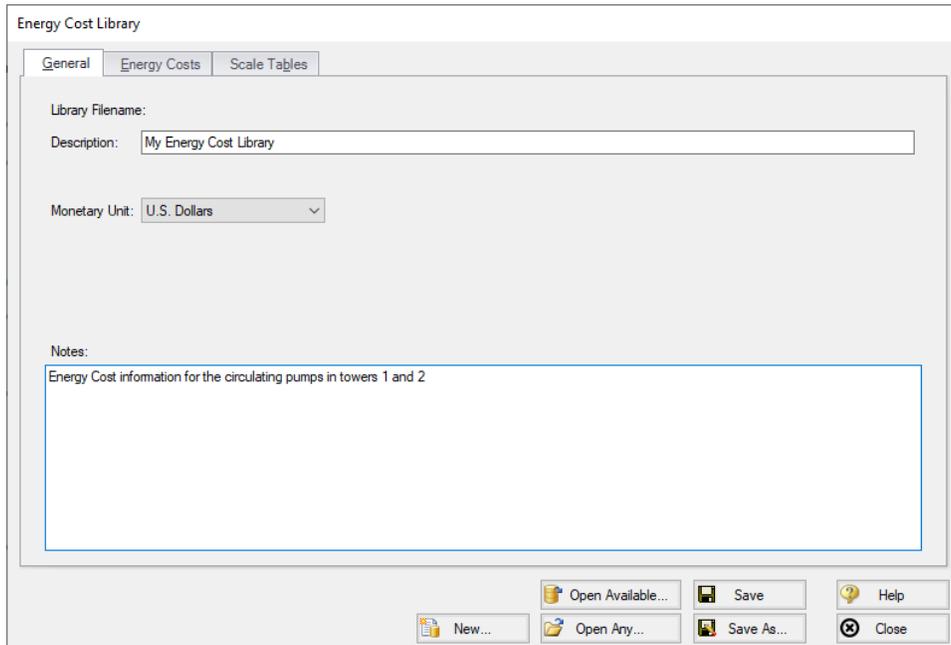


Figure 2: New Energy Cost Library

Defining Energy Costs

Once an Energy Cost Library is opened, the Energy Costs tab allows costs to be added. Select New Cost to add costs to the library. In Figure 3, costs have been added to represent a varying electricity cost over night and day. If the Cost Time Period exceeds the time defined in the Energy Cost Library, the pattern will start over at zero. That is, for the above example if the Cost Time Period was one year the defined 24 hour cost cycle would repeat 365 times.

The Cost Scaling column allows a Scale Table to be applied to a cost, as discussed below.

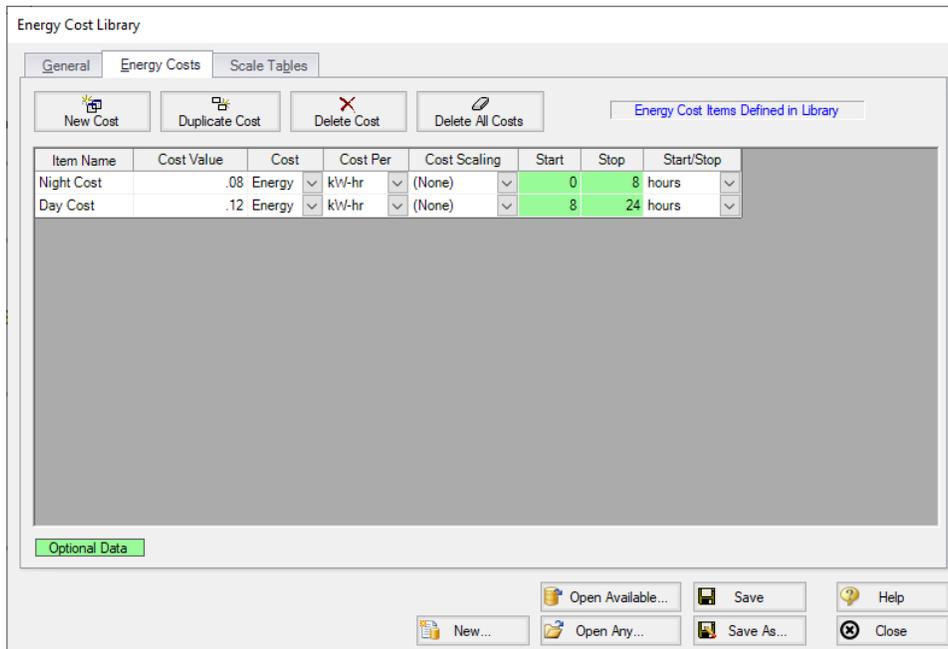


Figure 3: Energy Costs

Scale Tables

You can add a time-based multiplier to an Energy Cost Library on top of the normal Energy Costs if desired. The Scale Table can be used, for example, to represent the increasing cost of electricity year by year. See the [Scale Tables](#) topic for more information.

For energy cost libraries, the Scale Tables are limited to entering a multiplier over time.

Applying Energy Costs in the Model

The following must be completed to view the Operation/Energy column in the Cost Report:

- The [Cost Settings panel](#) in Analysis Setup must have the following definitions:
 - Cost Calculation set to either Pump Energy Only or Calculate
 - Cost Definition checkbox enabled for Operation/Energy
 - Energy Cost set to Use Energy Cost Libraries
 - Cost Time Period must have a non-zero System Life
- Pump junctions must have the appropriate selection from the [Library Jct](#) dropdown
- Cost tab of each Pump Properties window must be set to Include Cost in Report
- [Output Control](#) must have the Operation Costs parameter added under the Cost Report

Library Connection

Creating a new Energy Cost Library will prompt connecting it automatically. If for whatever reason an existing Library is not connected in can be manually connected with the [Library Manager](#).

Cost Application Manager

Note: This topic does not apply for the [ANS module](#). To set up cost calculations with ANS, see the [Assign Cost Libraries](#) topic.

The Cost Application Manager, accessed from an object's Cost tab, allows the user to do the following:

- Choose which cost libraries will be applied to each pipe and junction
 - Note that if multiple libraries are applied, costs from ALL of the selected libraries will be included.
- Choose when each pipe's or junction's life will start and/or end
- Specify fractional cost multipliers to be applied (for cases such as pump energy when the pump operates only part of the time)

The Cost Application Manager can also be accessed from the Tools menu - in which case it will show data for all Workspace objects.

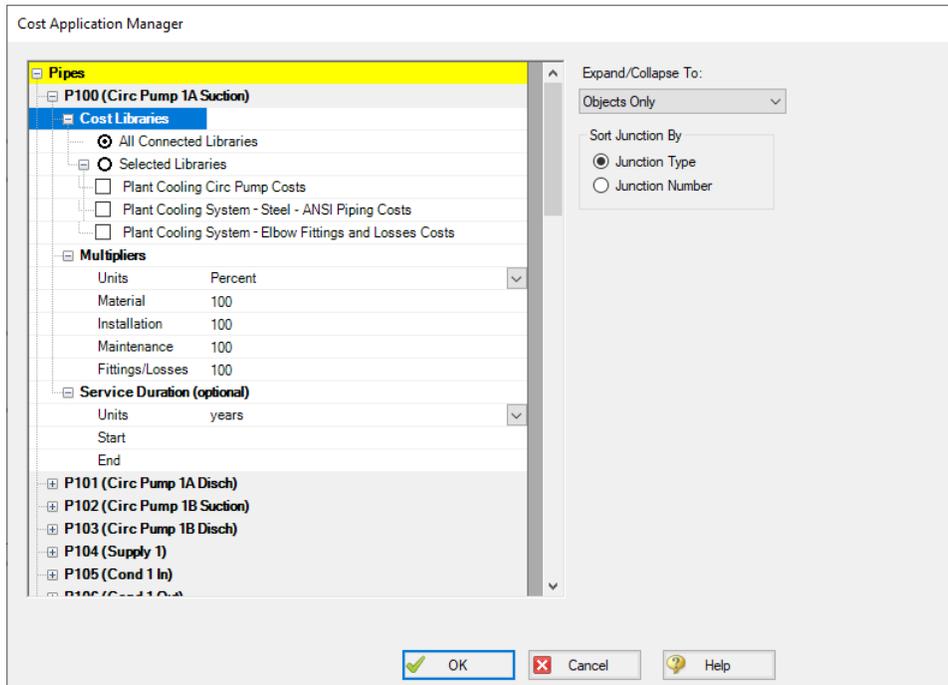


Figure 1: Cost Application Manager

How Duplicate Costs are Handled

To maximize flexibility for the user, multiple cost libraries can be connected to the same engineering library. What happens if there is cost data for the same engineering item in each library? For example, assume an engineering library, pump.dat, exists with a specific pump "PumpCo Model 100A".

Two cost libraries exist based on pump.dat:

- maxcost.cst ("PumpCo Model 100A" installation cost = \$1000)
- bestcost.cst ("PumpCo Model 100A" installation cost = \$750)

If both of these libraries are connected, and All Connected Libraries are applied to the pump, AFT Fathom will interpret the cost for "PumpCo Model 100A" as the sum of the two: \$1750.

There will be occasions you want cost items to be summed, and other cases where you do not. Obviously, if you have two libraries, one that tracks the maximum cost and the other that tracks the "best" cost of an item, you want to use only one cost at a time and not have the costs summed. This can be accomplished with the Cost Application Manager, by selecting only the desired library to apply.

However, perhaps the two acquisition costs do not represent the same item, but rather represent partial costs of that item that you want to maintain separately. For example, one library may contain costs of the pump and its casing, while another may contain cost for the motor. A third might contain cost for a variable speed drive. When managed in this way, you want the costs to be summed.

Related Topics

Cost Calculation 516
 Scale Tables 523
 Select Size From 525
 Cost Definitions 526
 Cost Libraries 529
 Cost Summary Window 533
 Custom Monetary Units 534
 Cost Report 535

Scale Tables

- Scale tables have a type (time or size) and a format (multiplier or cost). "Size" can refer to diameter, power, or C_v .
- Diameters can be quickly found and entered using the [Select Size From](#) button.
- The type of scale table defines the parameter that controls the variation in cost. For example, increasing diameter would generally cause an increase in cost, or energy prices may be projected to increase or decrease with time.
- Time type tables only accept the multiplier format.
- All scale tables require both a cost and a multiplier to determine the final cost value. Multiplier format tables require a base cost in the cost definition, cost format tables require a base multiplier.
- One scale table can be applied to multiple cost items - pipe materials, junctions, or fittings & losses.

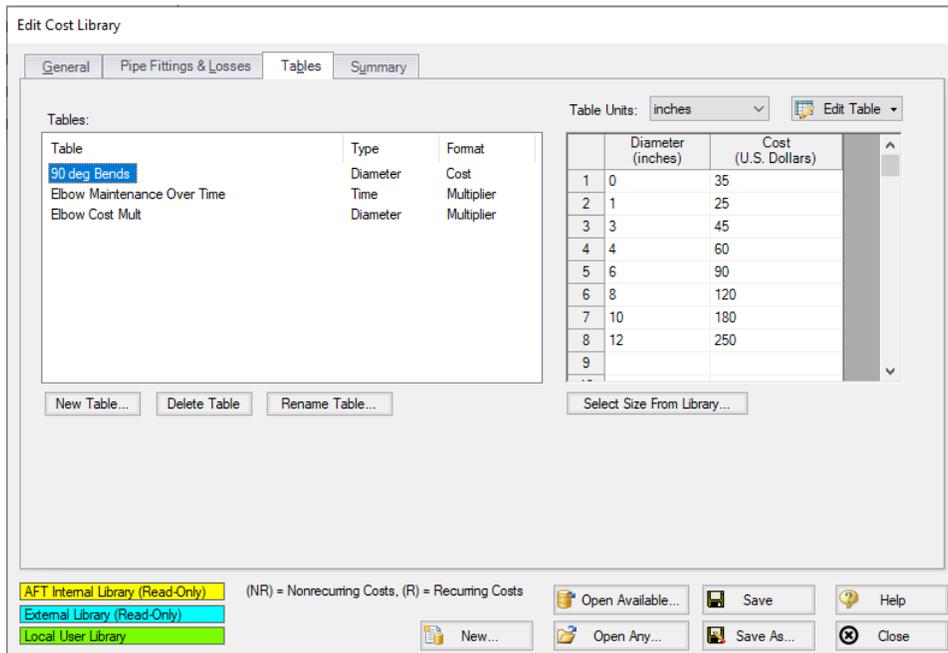


Figure 1: Scale table example of a table of costs (Diameter inches vs U.S. Dollars)

Energy Cost Libraries

Changes in [Energy Cost](#) over time can be captured with a scale table. Only time type scale tables can be applied to an Energy Cost Library.

Cost Libraries

Scale tables are applied in the cost definition area of the [Cost Library window](#). Pipe Materials only accept time scaling tables for recurring cost (shown in Figure 2)

	Cost #1	Cost #2	Cost #3
Description	1" Sch 40 Material	1" Sch 40 Installation	1" Sch 40 Maintenance
Cost Type	Material (NR) ▾	Installation (NR) ▾	Maintenance (R) ▾
Cost (U.S. Dollars)	2.2	7.2	
Cost Per	Length ▾	Length ▾	Time per Length ▾
Cost Per Units	feet ▾	feet ▾	years ▾
Cost Per Units #2			feet ▾
Time Scaling Table			Labor Inflation ▾

Figure 2: Time Scaling table applied to recurring cost

Fittings & losses and junction non-recurring costs can accept size scaling tables. Only one table can be applied to a cost item. The selection of a table of multipliers will require a base cost, and the selection of a table of costs requires a base multiplier (default of 1). This is demonstrated below.

	Cost #1	Cost #2
Material	Steel - ANSI ▾	Steel - ANSI ▾
Material Size	All Sizes ▾	All Sizes ▾
Material Type	All Types ▾	All Types ▾
Use Size Table	Table of Multipliers ▾	Table of Costs ▾
Cost (U.S. Dollars)	52	
Cost Per	Item ▾	
Cost Per Units		
Multiplier		1
Size Scaling Table	Elbow Cost Mult ▾	90 deg Bends ▾
Time Scaling Table		Low Maintenance Over Time ▾

Figure 3: Applying size tables to fittings & losses

Recurring costs can include both a size scaling table and a time scaling table, like in Figure 3. This would allow you to specify one base cost and use scale tables to represent both a range of sizes and an increasing cost over time, for example. Specifying both types of tables would be useful for a sizing calculation with the [ANS module](#), where cost is being minimized by varying the size.

Related Topics

Cost Calculation	516
Cost Application Manager.....	521
Select Size From.....	525
Cost Definitions.....	526
Cost Libraries.....	529
Cost Summary Window.....	533
Custom Monetary Units.....	534
Cost Report.....	535

Related Examples

Plant Cooling Cost Calculation	958
Cooling System - Creating Libraries - ANS	1157

Select Size From

The Select Size From button opens the Nominal Diameter window, which allows you to quickly obtain the inner diameter for a nominal size and schedule. The Diameter window is called from the [Scale Tables window](#).

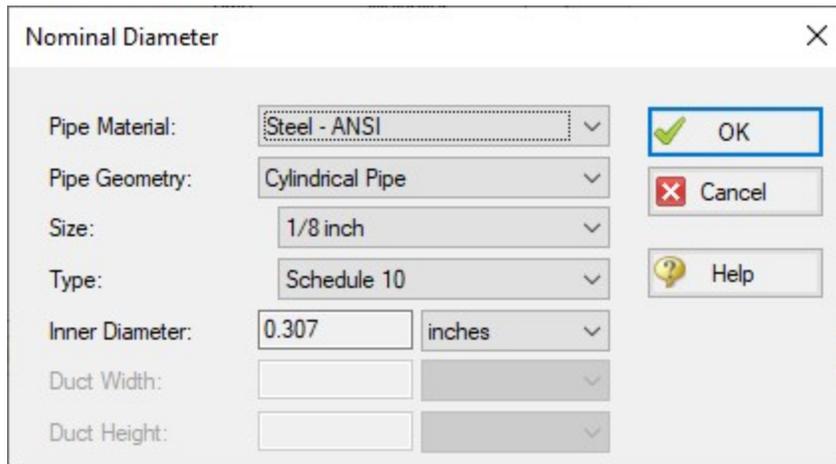


Figure 1: Nominal Diameter selection

Selecting a size and clicking OK enters the Inner Diameter into the currently selected Scale Table row.

Related Topics

Cost Calculation	516
Cost Application Manager.....	521
Scale Tables.....	523
Cost Definitions.....	526
Cost Libraries.....	529

Cost Summary Window.....	533
Custom Monetary Units.....	534
Cost Report.....	535

Cost Definitions

Depending on the contents of the associated Engineering Library there are tabs for Pipe Materials, Junctions, and Pipe Fittings & Losses. Each of these tabs serves the same purpose in slightly different ways - allowing the definition of costs for a certain component.

First, select a component of a certain type or size in the upper-left of the window. Then, click on the New Cost button. A new cost item is created in the table below as a new column. Some cells are displayed in black, which means that are not relevant to the type of entry selected.

Costs can be:

- Material (NR)
- Installation (NR)
- Maintenance (R)

where the (NR) and (R) indicate Non-Recurring and Recurring. Non-Recurring Costs are calculated as a sum total for the model, whereas Recurring costs require a time period and a total system life, as defined in the [Cost Settings panel](#) of Analysis Setup.

These costs are on either a per-item, per-length, or time basis. The costs can further be adjusted with [Scale Tables](#).

Pipe Materials Costs

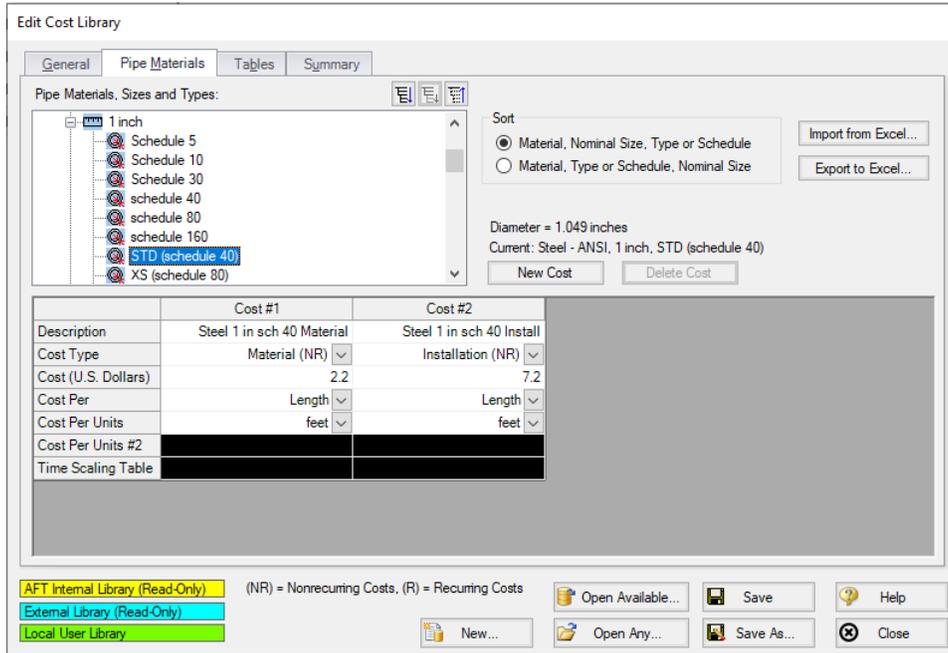


Figure 1: Cost Definitions for a Pipe Material

The Pipe Materials tab shows all the pipes in the engineering cost library. Costs can be entered at several levels. You can enter costs at the material level, the nominal size level, and finally at the type (e.g. schedule) level. Costs entered at the material apply to all pipes nominal sizes and types in that material type. Costs entered at the nominal size level apply to all schedules within that nominal size. Costs entered at the type level apply only to that type.

To make this process easier, values can be imported or exported to excel. The easiest way to import cost data from Excel is to first export the desired pipe Sizes and Types. Only the currently selected Sizes and Types will be exported (if the Pipe Material is selected, all will be exported) with any currently defined cost information included.

	A	B	C	D	E	F
1						
2	Cost Units:	U.S. Dollars				
3	Length Units:	feet				
4				===== COST PER UNIT LENGTH =====		
5	Material Name	Size	Type	Material	Installation	Maintenance
6	Copper Pipe - ASTM	1/8 inch	STD			
7	Copper Pipe - ASTM	1/8 inch	XS			
8	Copper Pipe - ASTM	1/4 inch	STD			

Figure 2: Excel Sheet for Importing Cost Definitions

The empty columns for cost per unit length can be filled in where cost data is required - it is not necessary to supply data for every cost, or for every Size/Type. After saving this sheet, the data can be imported directly with "Import from Excel."

Recurring costs for Pipe Materials can be modified with a Time [Scaling Table](#).

Junctions Costs

Costs can be entered for all junction types except the volume balance.

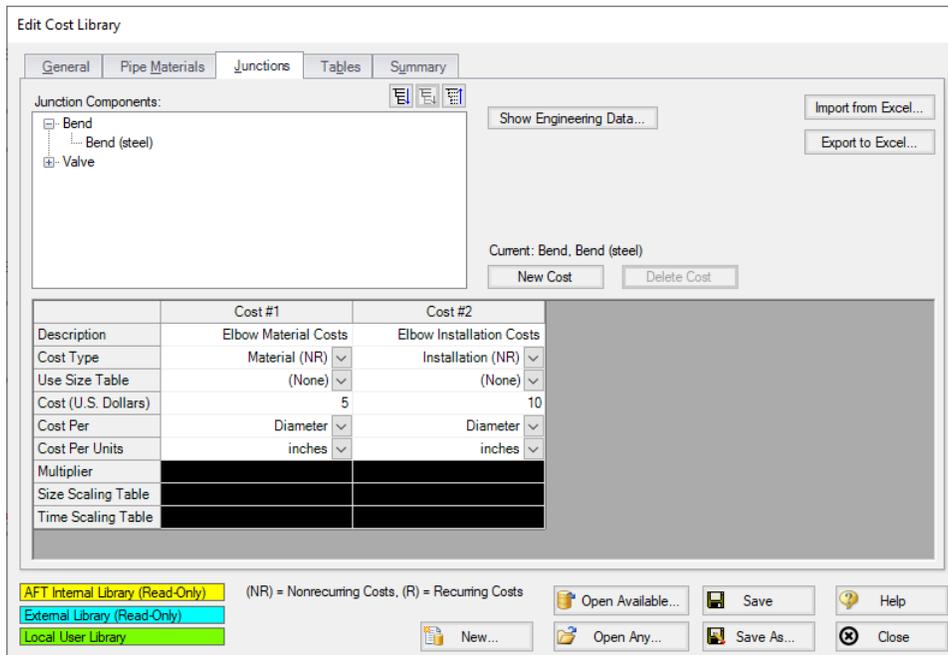


Figure 3: Cost Definitions for a Bend

Above is an example of entering cost data for a bend junction. Junction costs can also be entered at a higher level. For example, you can apply a cost to all Smooth Flanged Elbows.

Junction costs can be scaled with a Table of Multipliers, Table of Costs, or a Time Scaling Table.

The behavior of certain junctions varies slightly:

- **Pumps** - Cost can be entered per item (pump) or per Power. This will allow an approximate scaling for the size of the pump on a more reasonable basis than diameter.
- **Control Valves** - Costs can be entered per item or diameter. With a Scale Table, costs can be scaled based on C_v .
- **Other Junctions** - Costs can be entered per item or diameter. Scaling tables can also be used on a diameter basis.

Pipe Fittings & Losses Costs

Cost specification functions the same as non-pump junctions, with the exception that fitting & loss costs can be associated with a particular pipe material and size. Since the pipe fitting & loss engineering library is included only in the AFT Fathom internal library, cost data for pipe fittings & losses can only be entered when the cost library is associated with the internal engineering library – unless you add your own custom fittings & losses to an engineering library. Then cost libraries can be associated with that library as well.

Related Topics

Cost Calculation.....	516
Cost Application Manager.....	521
Scale Tables.....	523
Select Size From.....	525
Cost Libraries.....	529
Cost Summary Window.....	533
Custom Monetary Units.....	534
Cost Report.....	535

Related Examples

Beginner - Cost Calculation.....	908
Plant Cooling Cost Calculation.....	958

Cost Libraries

AFT Fathom can track user-defined costs such as maintenance, installation and material cost with a Cost Library.

Note that AFT Fathom is not designed as a cost estimator. Instead, user-defined cost data can be used to compare different designs to determine which one has a lower cost. Like all parts of the model, the results of a cost calculation depend on how the costs have been defined by the user.

Relationship to Engineering Libraries

AFT Fathom uses Engineering Libraries to create collections of standardized components.

Cost Libraries must be created based on an engineering library to provide user-defined cost values for components in the engineering library. There can be [multiple cost libraries](#) for each engineering library. This allows cost data sets to be maintained separately from the engineering data. This can be useful for maintaining costs in different currencies or differences based on geographical location or times of the year.

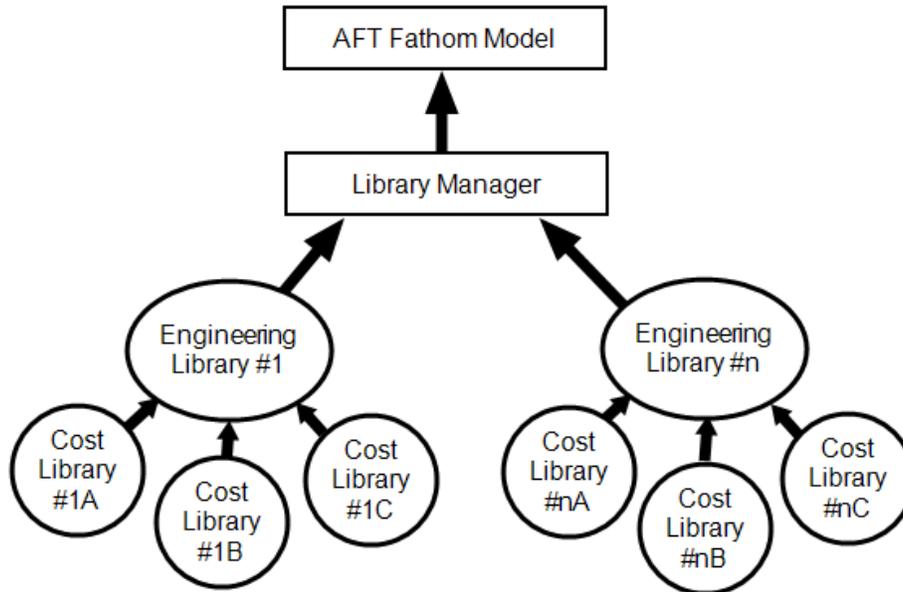


Figure 1: Relationship between Cost and Engineering Libraries

Like Engineering Libraries, Cost Libraries must be connected to be used. See [Library Manager](#) for information on how to connect libraries. Newly created cost libraries are connected automatically.

Edit Cost Library Window

The Edit Cost Library window is opened from the Library menu. This window allows you to create a cost library for items that are in a specific [engineering library](#). Costs can be entered for pipes, junctions, and pipe fittings & losses.

On the General tab, a cost library can be created with the New button, or an existing library can be accessed with one of the Open buttons.

Cost Calculation

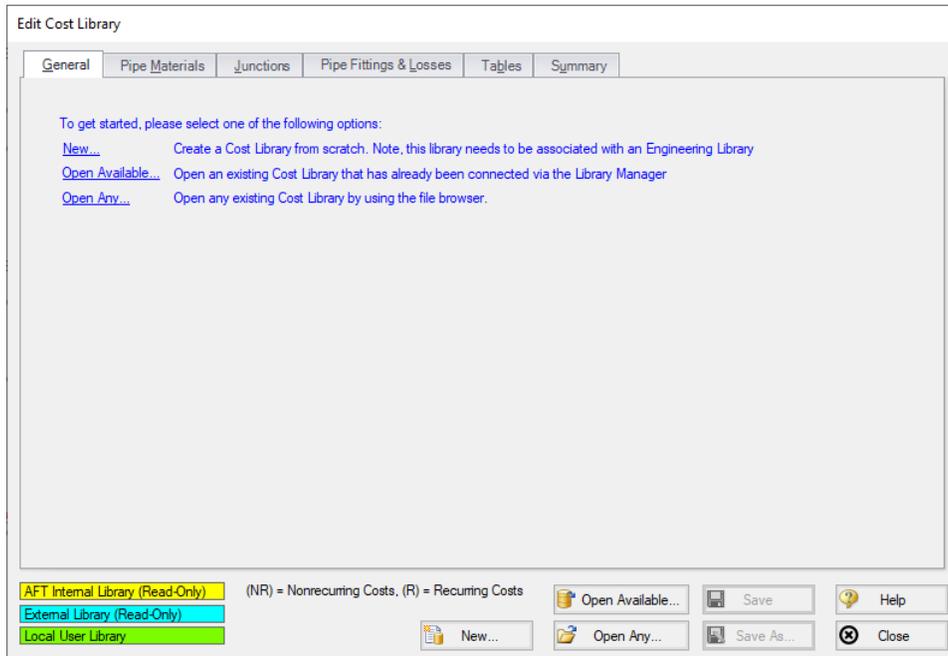


Figure 2: General tab of Edit Cost Library Window

Clicking New brings up the following window:

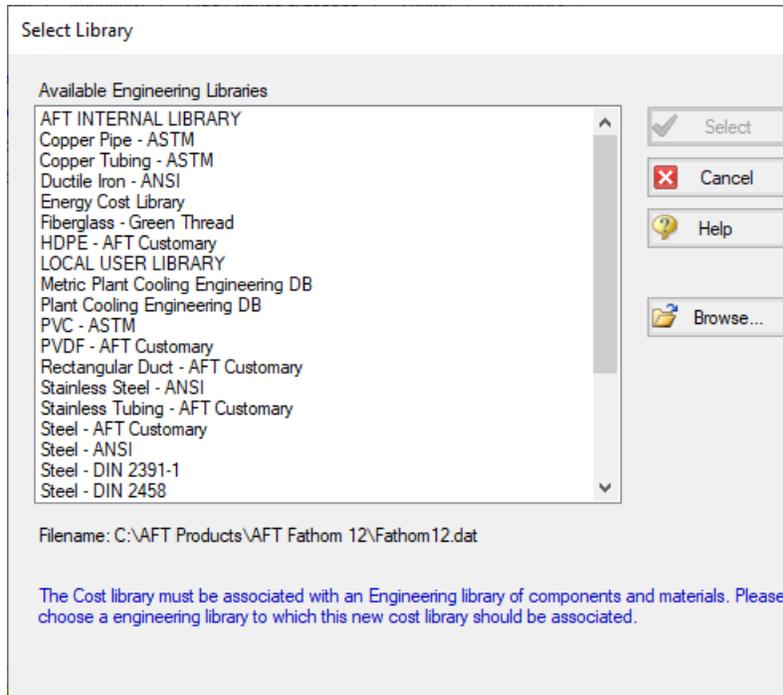


Figure 3: Select Library Window

All Cost Libraries need to be associated with an Engineering Library. After selecting a library, the Edit Cost Library Window will change to allow you to enter information into the newly created Cost Library.

The screenshot shows the 'Edit Cost Library' window with the 'General' tab selected. The fields are as follows:

- Cost Library Filename: None
- Engineering Library Filename: C:\AFT Products\AFT Fathom 12\Pipe Material Libraries\Steel - ANSI.dat
- Engineering Library Description: Steel - ANSI
- Cost Type: Monetary Non-Monetary
- Monetary Unit: U.S. Dollars
- Description: (empty text box)
- Notes: (empty text area)

The 'Multipliers on Cost Items' table is shown below:

Cost Items	Material	Installation	Maintenance
Pipe Materials	1	1	1
Junctions	1	1	1
Fittings/Losses	1	1	1

At the bottom of the window, there is a 'Set All Multipliers to One' button and a list of libraries: AFT Internal Library (Read-Only), External Library (Read-Only), and Local User Library. The bottom right contains buttons for 'Open Available...', 'Save', 'Help', 'New...', 'Open Any...', 'Save As...', and 'Close'.

Figure 4: Edit Cost Library Window - New Library

A Description is required for all Cost Libraries, which is displayed as the library's name in the Library Manager. Multipliers can be set for all items in the cost library using the Multipliers on Cost Items table. Notes are optional.

Cost Definitions

Depending on the contents of the associated Engineering Library there are tabs for Pipe Materials, Junctions, and Pipe Fittings & Losses. Each of these tabs serves the same purpose in slightly different ways - allowing the definition of costs for a certain component. See [Cost Definitions](#) for a detailed explanation.

Summary

The summary tab allows for an easy way to quickly review what has been defined in a cost library. This is different than the [Cost Summary Window](#), as this summary is a listing of all items defined as costs, whereas the Cost Summary Window lists objects in the model with costs applied to them.

The costs are grouped by type, and only defined costs are shown. An example of defined costs on a Pipe Materials library is shown below.

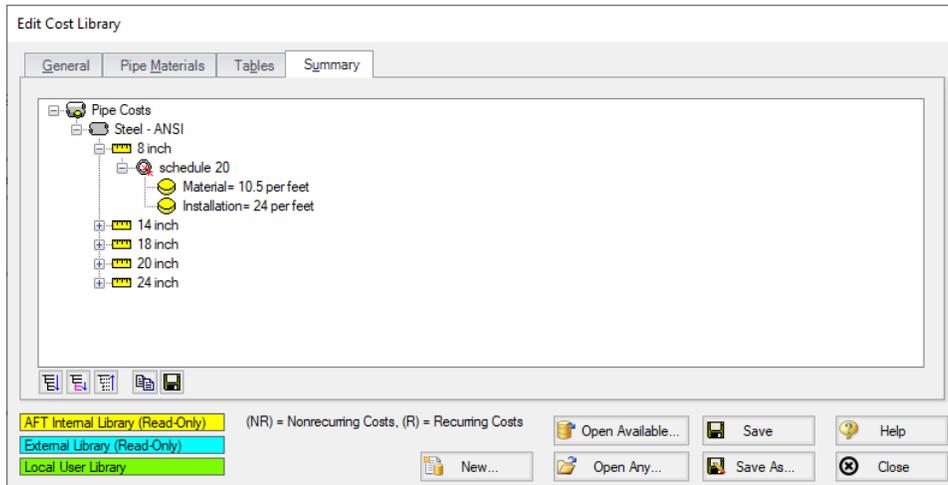


Figure 5: Cost Summary

Related Topics

Cost Calculation	516
Cost Application Manager	521
Scale Tables	523
Select Size From	525
Cost Definitions	526
Cost Summary Window	533
Custom Monetary Units	534
Cost Report	535

Related Examples

Beginner - Cost Calculation	908
Plant Cooling Cost Calculation	958
Cooling System - Creating Libraries - ANS	1157

Cost Summary Window

Note: This topic does not apply for the [ANS module](#). To set up cost calculations with ANS, see the [Assign Cost Libraries](#) topic.

The Cost Summary window, opened from the Tools menu shows all connected [cost libraries](#) and [energy cost libraries](#), and which pipes and/or junctions are obtaining costs from each library.

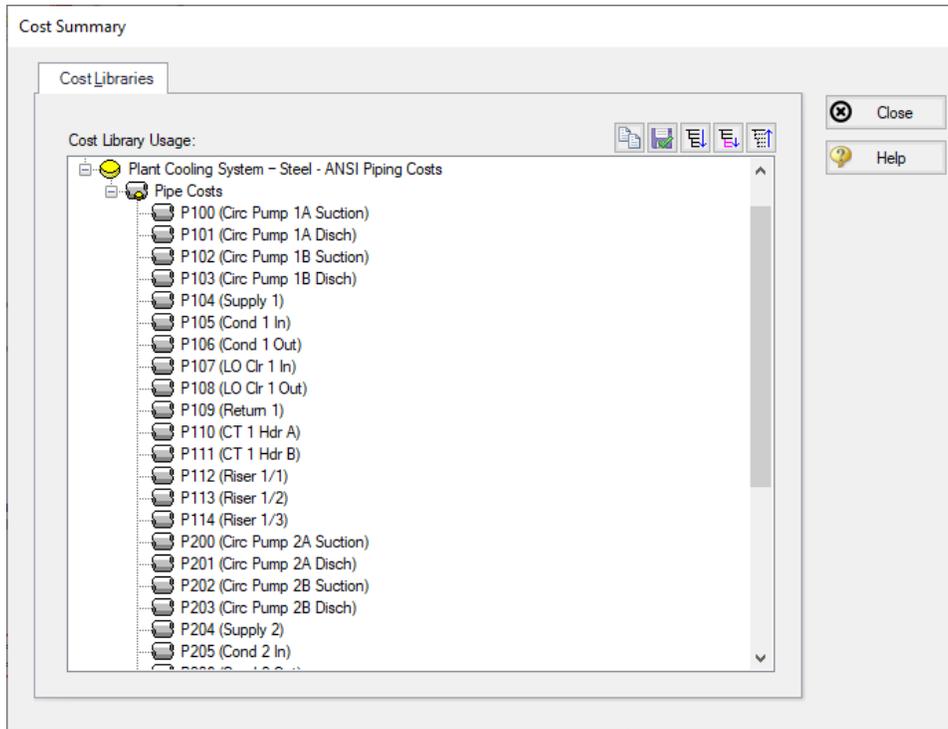


Figure 1: Cost Summary Window

Related Topics

Cost Calculation	516
Cost Application Manager	521
Scale Tables	523
Select Size From	525
Cost Definitions	526
Cost Libraries	529
Custom Monetary Units	534
Cost Report	535

Custom Monetary Units

New Monetary Units can be added with the following steps:

1. Open [User Options](#) from the Tools menu
2. Under Unit Systems, select Show/Hide Unit Items
3. In the Unit Family Type list box, select Monetary Units
4. The Custom Units button will appear on the right, select New
5. Enter the name, description and conversion to U.S. Dollar in the appropriate fields.
6. Click OK

The new Monetary units can be specified as your default units and used in [Cost Libraries](#). If you do have a mixture of cost libraries with different Monetary Units, the conversion factors to U.S. Dollars will need to be updated before the cost is calculated.

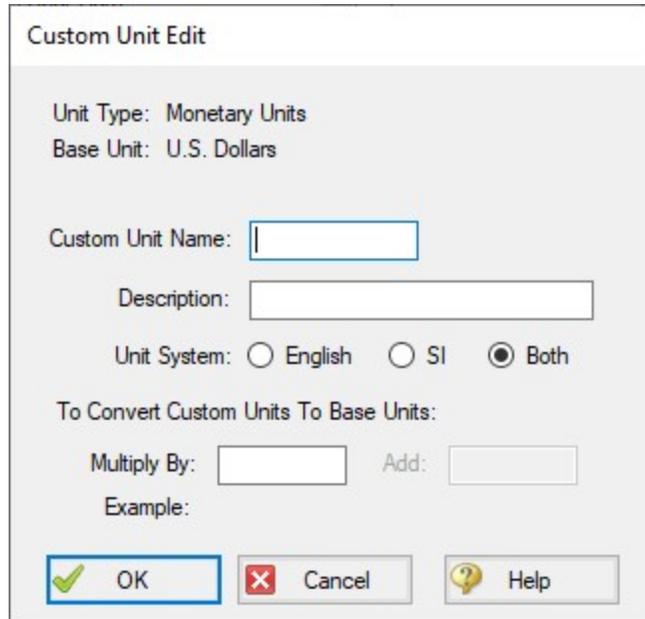


Figure 1: Custom Unit Edit Window

Related Topics

Cost Calculation.....	516
Cost Application Manager.....	521
Scale Tables.....	523
Select Size From.....	525
Cost Definitions.....	526
Cost Libraries.....	529
Cost Summary Window.....	533
Cost Report.....	535

Cost Report

The Cost Report is a detailed list of all costs for the system displayed on the Cost Report tab in the General section of the [Output window](#). The content of the Cost Report can be customized in the [Output Control](#) window on the Display Parameters tab and the Format & Actions tab.

The grand total cost is shown on the top line, with subtotals for each category on the second line. In the main table area costs for each pipe, junction, and pipe fitting & loss included in the cost calculation are shown.

Output Control Format & Action allows you to hide items that have zero cost, group costs by pipe, or show only the subtotals for each category. You can also edit the number of decimal places shown, or

Cost Calculation

adjust the costs to be truncated to be shown in terms of hundreds, thousands, etc. instead of showing the full number.

	Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/ Energy	Recurring Sub Total	TOTAL
TOTAL OF ALL MODEL COSTS										3,170,612
Total of All Shown Costs				405,480	677,460	1,082,940	0	2,087,672	2,087,672	3,170,612
Pipe Subtotal				291,480	627,460	918,940	0	0	0	918,940
P100		Pipe	Circ Pump 1A Suction	354	855	1,209	0	0	0	1,209
P101		Pipe	Circ Pump 1A Disch	708	1,710	2,418	0	0	0	2,418
P102		Pipe	Circ Pump 1B Suction	354	855	1,209	0	0	0	1,209
P103		Pipe	Circ Pump 1B Disch	708	1,710	2,418	0	0	0	2,418
P104		Pipe	Supply 1	60,000	129,000	189,000	0	0	0	189,000
P105		Pipe	Cond 1 In	2,900	5,500	8,400	0	0	0	8,400
P106		Pipe	Cond 1 Out	2,900	5,500	8,400	0	0	0	8,400
P107		Pipe	LO Clr 1 In	210	480	690	0	0	0	690

Figure 1: The Cost Report is displayed in the General section of the Output window

Using the Library Sources Tables

AFT Fathom displays the source of all cost data in a run, and you are strongly encouraged to review this data to verify that the solution was based on the appropriate cost data.

In the [Output window](#), the sources of all cost data are displayed for each pipe and junction in the Library Sources table. Figure 2 shows the Library Sources for an example case.

Pipe	Name	Pipe Material	Pipe Installation	Fittings Material	Fittings Installation
100	Circ Pump 1A Suction	Plant Cooling System - Steel - ANSI Piping Costs	Plant Cooling System - Steel - ANSI Piping Costs		
101	Circ Pump 1A Disch	Plant Cooling System - Steel - ANSI Piping Costs	Plant Cooling System - Steel - ANSI Piping Costs		
102	Circ Pump 1B Suction	Plant Cooling System - Steel - ANSI Piping Costs	Plant Cooling System - Steel - ANSI Piping Costs		
103	Circ Pump 1B Disch	Plant Cooling System - Steel - ANSI Piping Costs	Plant Cooling System - Steel - ANSI Piping Costs		
104	Supply 1	Plant Cooling System - Steel - ANSI Piping Costs	Plant Cooling System - Steel - ANSI Piping Costs		
105	Cond 1 In	Plant Cooling System - Steel - ANSI Piping Costs	Plant Cooling System - Steel - ANSI Piping Costs		
106	Cond 1 Out	Plant Cooling System - Steel - ANSI Piping Costs	Plant Cooling System - Steel - ANSI Piping Costs		
107	LO Clr 1 In	Plant Cooling System - Steel - ANSI Piping Costs	Plant Cooling System - Steel - ANSI Piping Costs		
108	LO Clr 1 Out	Plant Cooling System - Steel - ANSI Piping Costs	Plant Cooling System - Steel - ANSI Piping Costs		
109	Return 1	Plant Cooling System - Steel - ANSI Piping Costs	Plant Cooling System - Steel - ANSI Piping Costs		
110	CT 1 Hdr A	Plant Cooling System - Steel - ANSI Piping Costs	Plant Cooling System - Steel - ANSI Piping Costs		

Figure 2: Library Sources table for pipes

Related Examples

Beginner - Cost Calculation	908
Plant Cooling Cost Calculation	958
Cooling System - ANS	1176

Related Topics

Cost Calculation	516
Cost Application Manager	521
Scale Tables	523
Select Size From	525
Cost Definitions	526
Cost Libraries	529
Cost Summary Window	533
Custom Monetary Units	534

Add-on Modules

AFT Fathom has four optional add-on modules which extend its modeling capabilities into other areas. The modules allow multivariable goal seeking and control simulation, settling slurry modeling, time varying simulation, and full system sizing. The [Modules panel](#) within the Modules group in [Analysis Setup](#) allows you to activate and enable any of the four AFT Fathom add-on modules. An additional license to the base AFT Fathom license is required to use a module. These modules include:

- [GSC \(Goal Seek & Control\)](#) - Identifies input parameters that yield desired output values and simulates control functions.
- [SSL \(Settling Slurry\)](#) - Models the effects of pumping fluids containing settling solids using the Wilson/GIW method.
- [XTS \(Extended Time Simulation\)](#) - Models dynamic system behavior and how critical system parameters vary over time.
- [ANS \(Automated Network Sizing\)](#) - Evaluates the complex interaction of system variables to reveal combinations of cost savings.

Related Topics

Modules Group.....	84
Modules Panel.....	85

GSC - Goal Seek & Control

What is the GSC Module?

An optional add-on module to AFT Fathom which allows users to perform multi-variable goal seeking and simulate system control functions.

Many engineering modeling tasks involve more than just directly solving a system. In some cases a manual, “cut and try” method of changing input variables to achieve desired operating results is required. When a single parameter is being changed, the manual process (while tedious and time consuming) can be used successfully. However, varying two or more parameters at the same time quickly becomes impractical.

The GSC module automates the process of changing input variables to achieve desired design goals. For single variable situations, the GSC module offers the advantage of being much faster than manual methods. Further, it provides a practical tool for solving cases when there are two or more variables.

How Does the GSC Module Work?

The GSC module employs numerical optimization technology like that used in the [AFT Fathom ANS Module](#). The optimization engine employed by the AFT Fathom GSC module uses state-of-the-art optimization technology licensed from Vanderplaats Research and Development, the leading company in

optimization technology. However, rather than minimizing some function value such as system cost or weight, the GSC module satisfies equality criteria using “goal programming” methods.

The core hydraulic solution in AFT Fathom is performed by the Hydraulic Solver. In standard AFT Fathom usage, the Hydraulic Solver is called once to solve a system. The GSC module’s Numerical Optimizer is used to call the Hydraulic Solver repeatedly, thus solving a series of models with different inputs. The Numerical Optimizer adjusts the user specified Variables in order force the Hydraulic Solver output values to agree with the user specified Goals. Figure 1 depicts the relationship between the different components.

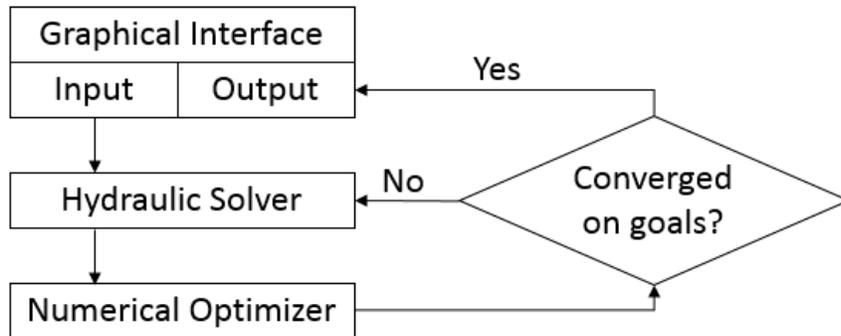


Figure 1: The GSC module flowchart shows how the Hydraulic Solver is called repeatedly in an iterative loop

Using the GSC Module

The user has the option of activating or not activating the GSC module when AFT Fathom first loads. After AFT Fathom is loaded, the GSC module can be activated or deactivated for use from the Modules panel in the Analysis Setup window or the Tools menu. Whether or not GSC is activated impacts the View menu, Analysis menu, Model Data window, Solution Progress window, and Output window as is discussed in the Modules panel topic.

If the GSC module is active, the user can still run models without goal seeking. This is selected under Goal Seek & Control on the Analysis menu. Hence there are three possibilities for GSC.

- GSC is not active
- GSC is active and goal seeking is ignored
- GSC is active and goal seeking is performed

Related Topics

Modules Panel.....	85
Add-on Modules.....	538
Goal Seek and Control Group.....	148

Variables Panel.....	149
Goals Panel.....	155
Numerical Controls Panel.....	161
GSC in Model Data.....	540
GSC in Solution Progress.....	542
GSC in Output.....	543
GSC in Scenarios.....	543
Changing Input Values to GSC Results.....	544
GSC Troubleshooting.....	545

Related Examples

Beginner - Heat Transfer in a Pipe - GSC.....	994
Spray Discharge System - GSC.....	1019
Pump Sizing and Selection with Flow Control Valves - GSC.....	1001
Cooling System - GSC.....	1006
Controlled Heat Exchanger Temperature - GSC.....	1027
Model Calibration with Pipe Variables - GSC.....	1011
NFPA Firewater System - GSC.....	1032
Fixed Head Supply Tank - GSC XTS.....	1263

GSC in Model Data

([GSC Module Only](#)) When the GSC module is active, the [Model Data window](#) displays a list of all variables and goals in the General section. This is useful for documentation purposes. See Figure 1.

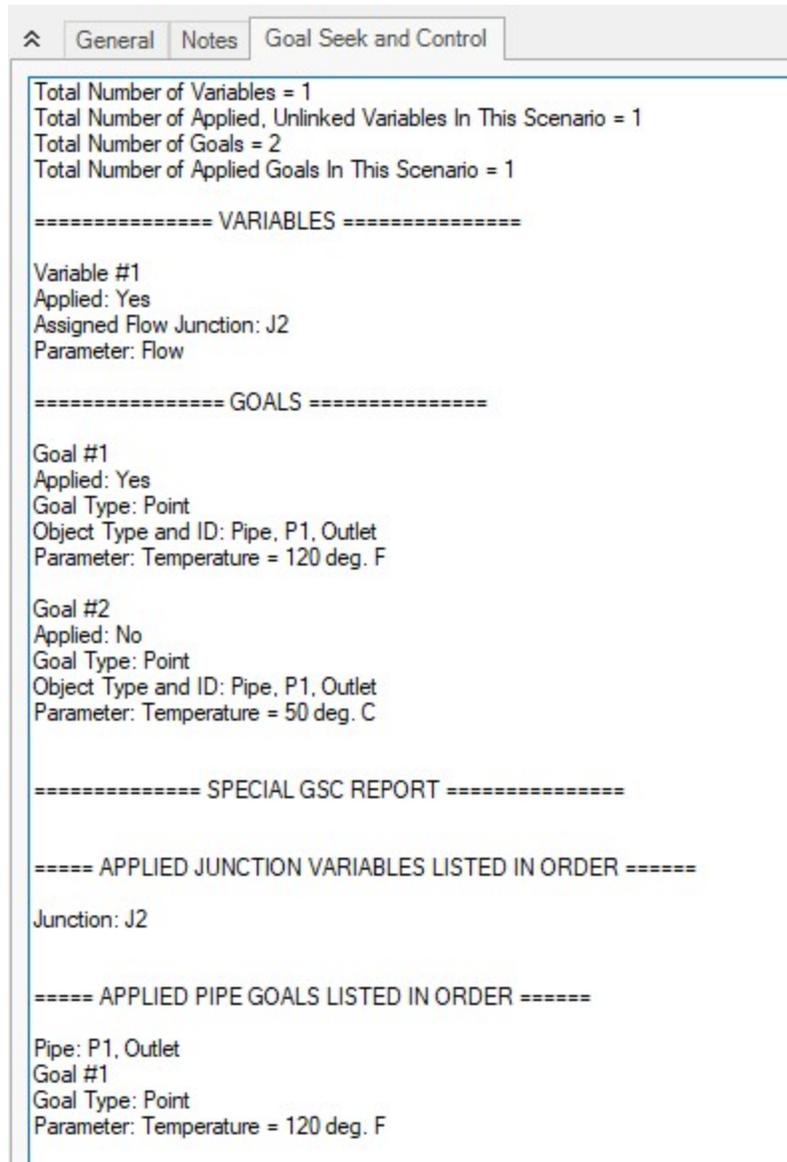


Figure 1: Model Data window shows the variable and goal input data

Related Topics

GSC - Goal Seek & Control.....	538
GSC in Solution Progress.....	542
GSC in Output.....	543
GSC in Scenarios.....	543
Changing Input Values to GSC Results.....	544
GSC Troubleshooting.....	545

GSC in Solution Progress

[\(GSC Module Only\)](#) When a model is run with GSC active, an additional line will be shown in the Solution Progress window to indicate the progress that the GSC module has made towards reaching the specified goal. There are three fields displayed, as can be seen in Figure 1 below.

The "Best (Lowest)" and "Current" fields indicate the relative difference between the GSC iterations and the goal. The "Current" field shows the difference for the current iteration, and the "Best" field tracks the difference for the iteration that had the smallest difference from the goal. Both fields start at 100, as the initial iteration is the farthest from the goal. Both fields will decrease in magnitude as GSC proceeds closer to reaching the goal value. If these numbers are not significantly changing when the model is run, this is an indication that GSC may not be able to reach the goal with the current model settings. See the [Goal Seek and Control Troubleshooting](#) topic for more information on resolving this.

The field on the far right of the GSC Calls to Solver line tracks the total iterations performed by GSC. For every GSC iteration, GSC makes a call to the hydraulic solver in Fathom to determine the hydraulic solution with the adjusted value of the GSC variable. Thus, this value will always be smaller than the flow and head iterations.

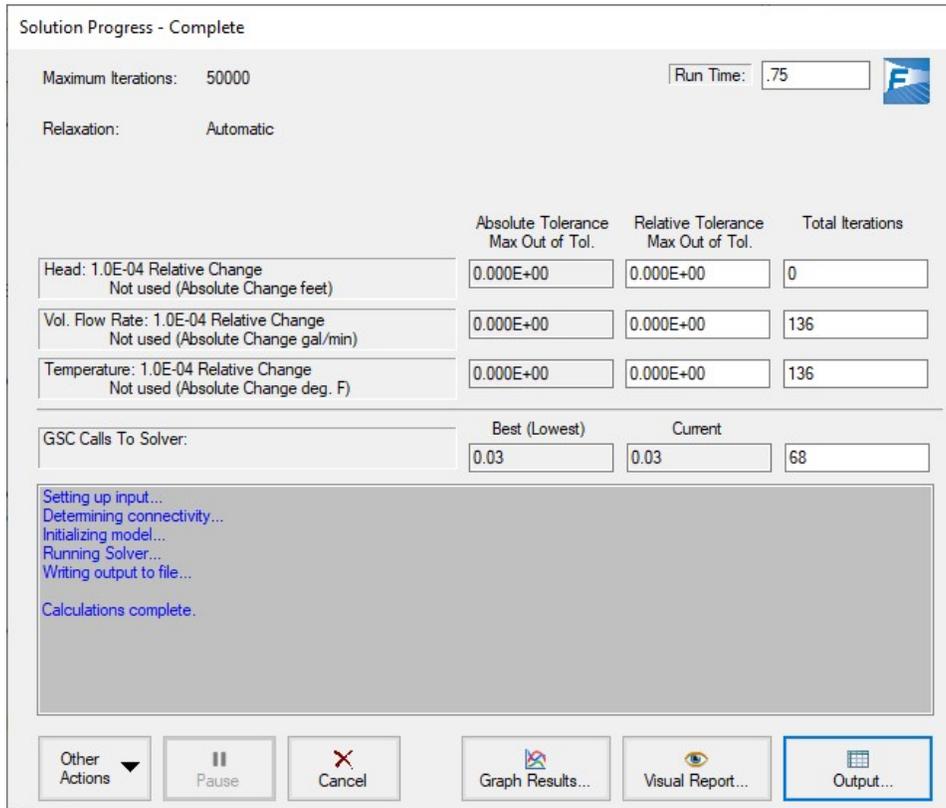


Figure 1: The Solution Progress window has an additional line to show information on the GSC Calls to Solver when the Fathom model is run with GSC enabled

Related Topics

GSC - Goal Seek & Control..... 538
 GSC in Model Data..... 540
 GSC in Output..... 543
 GSC in Scenarios..... 543
 Changing Input Values to GSC Results..... 544
 GSC Troubleshooting..... 545

GSC in Output

([GSC Module Only](#)) The results for a Goal Seek & Control run are displayed in the [Output window](#) in the general section in two tables called GSC Variables and GSC Goals (see Figure 1).

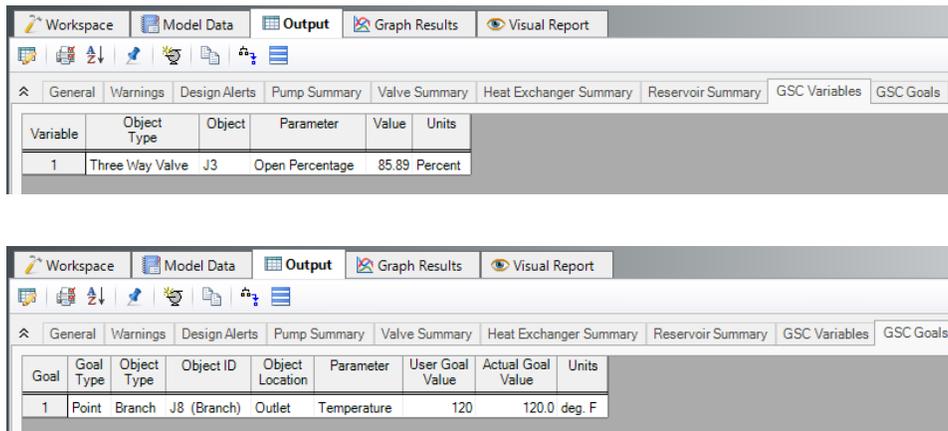


Figure 1: Calculated GSC Variables table (top) shows values required to achieve goals. GSC Goals table (bottom) shows user’s goals and actual values achieved (these should match).

Related Topics

GSC - Goal Seek & Control..... 538
 GSC in Model Data..... 540
 GSC in Solution Progress..... 542
 GSC in Scenarios..... 543
 Changing Input Values to GSC Results..... 544
 GSC Troubleshooting..... 545

GSC in Scenarios

GSC [variables](#) and [goals](#) will appear in all [scenarios](#). In other words, you cannot create a variable or goal that will appear in some scenarios and not others. Deleting a variable or goal in one scenario will delete it in all scenarios. However, the application of variables and goals can differ from scenario to scenario. This allows you to perform goal seeking with different variables and goals being used in different scenarios.

Related Topics

GSC - Goal Seek & Control.....	538
GSC in Model Data.....	540
GSC in Solution Progress.....	542
GSC in Output.....	543
Changing Input Values to GSC Results.....	544
GSC Troubleshooting.....	545

Changing Input Values to GSC Results

([GSC Module Only](#)) As discussed previously, the Output window [GSC Variables tab](#) indicates how the input parameters must be varied to satisfy the goals. Once you have these results, you may want to keep and use them in the model. There are two ways to do this: [Transferring Results to Initial Guesses](#) and generating a [Disconnected Scenario](#).

Transferring Results to Initial Guesses

In general, the Output window supports a feature called Transfer Results to Initial Guesses. This is available on the [Edit Menu](#). When used without the GSC module, it transfers calculated results for flow, pressure and temperature to the initial guess values of the pipes and junctions. When used after a GSC run it gives you three options: to transfer the GSC Variable Data, the calculated results, or both.

If you choose either of the first two options that transfer GSC Variable Data, the junction data on the Properties Window will be changed to the GSC Variable results. This means that one could then choose to Ignore goal seeking, rerun the model, and obtain the same results as before. The options to transfer the calculated results will transfer the pipe and junction results to the initial guesses field, similar to what would be done if the GSC module were not enabled.

Generating a Disconnected Scenario

Generating a [disconnected scenario](#) is ideal for GSC results in that it may not make sense to save the GSC results for future use if the model can otherwise be changed in arbitrary ways - if the model were changed the GSC results would be invalidated, meaning the user would need to rerun GSC.

For example, assume a GSC run is made to find the valve open position required to achieve a certain flow. If this result is saved, and then the pipe size is later changed, the valve open position would no longer supply the desired flow.

Creating a disconnected scenario helps prevent changes to the scenario since disconnected scenarios cannot be changed by propagated changes from other scenarios.

Related Topics

GSC - Goal Seek & Control.....	538
GSC in Model Data.....	540
GSC in Solution Progress.....	542
GSC in Output.....	543

GSC in Scenarios.....	543
GSC Troubleshooting.....	545

GSC Troubleshooting

([GSC Module Only](#)) See the below topics for more information on troubleshooting issues with the GSC module:

- [When Goals Cannot Be Achieved](#)
- [Goal Seek and Control Adjustments](#)
- [What To Do if the Solver Gets Stuck in GSC](#)
- [Goal Seek and Control Inconsistencies](#)

Related Topics

GSC - Goal Seek & Control.....	538
GSC in Model Data.....	540
GSC in Solution Progress.....	542
GSC in Output.....	543
GSC in Scenarios.....	543
Changing Input Values to GSC Results.....	544

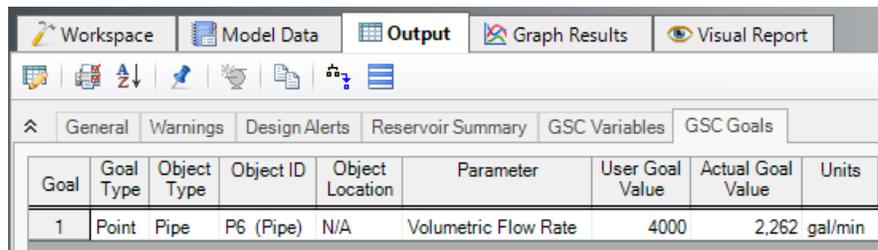
When Goals Cannot Be Achieved

([GSC Module Only](#)) In some cases the GSC module may not be able to achieve a [goal](#). This can happen for the following reasons:

- A goal is not physically realistic
- A better starting point is needed for the input values assigned as GSC [Variables](#)
- Hydraulic solution tolerances or optimizer numerical parameters require adjustment

In such cases, the goal will not be satisfied in the results. In other words, the Actual Goal Value and User Goal Value will differ (Figure 1).

In all cases the user will be warned when goals were not met. Figure 1 shows the Output window with such a warning.



Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units
1	Point	Pipe	P6 (Pipe)	N/A	Volumetric Flow Rate	4000	2,262	gal/min



Messages

For an explanation of Warnings, click an item in this list and press F1 or search for 'Warnings' in the Help system.

- WARNING**
GSC Goal may not be valid. User goal: 4000 gal/min, Final goal result: 2,262.14 gal/min.
- RECOMMENDATION**
It is recommended that Solution Control tolerances or Goal Seek and Control numerical parameters be adjusted. Press the F2 key to automatically adjust parameters.

Figure 1: Warning display in Output window when a goal cannot be met

Physically Unrealistic Goals

It may be possible that a goal is not achievable because it is not physically possible. For example, a pump may be too small to satisfy a flow rate goal at some pipe.

To determine whether a goal is unrealistic, try changing the goal and rerunning the model. If the goal can be met in some cases and not others, this is an indicator that the original goal was unrealistic.

Better Starting Point Needed

In some cases the goal may be so far away from the original model that the GSC module has difficulty finding a solution. This is not common, but it is worth trying to adjust the starting point of parameters which are being varied.

Changing Control Parameters

The Numerical Optimizer determines goal search directions by approximating a gradient based on perturbed variable values. By default it uses a central difference approximation, but forward difference can also be used. If the hydraulic solution is not converged to a sufficiently small tolerance, the gradient approximation will not be sufficiently accurate and hence a good search direction cannot be obtained. The end result is that the goal seeking fails.

There are two general area that can be changed in such cases: the hydraulic solution tolerances in the [Steady Solution Control group](#) and [Numerical Controls panel](#) in [Analysis Setup](#).

In the next few sections recommendations on how to manually change these parameters will be given. However, before making a manual change it is worthwhile to try the automated parameter adjustment feature.

Figure 1, bottom picture, shows the warning when a goal is not met, and also shows a recommendation and suggest pressing the F2 key. When doing so, a window appears as shown in Figure 2. This window will offer up to three options for automatic parameter adjustment. Accepting the default values is typically a good starting point. Click OK, and rerun the model.

The purpose of the window shown in Figure 2 is to simplify making common changes to resolve goal seeking problems.

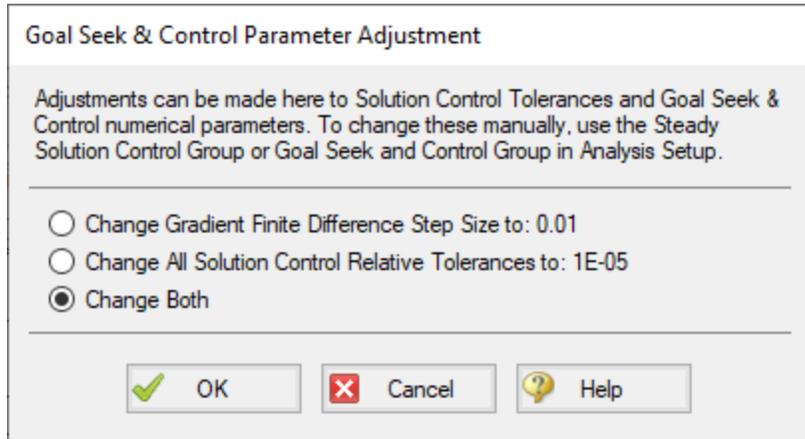


Figure 2: When goal seeking fails, pressing the F2 key will show this window which allows the user to automatically implement recommend numerical parameter adjustments

Caution: The user should be very cautious in manually changing parameters in the [Steady Solution Control Group](#) and [Numerical Controls panel](#). Making changes without a good understanding of the parameters involved can lead to incorrect results.

- **Changing Tolerance on Tolerance panel** - By default AFT Fathom uses Relative Tolerance criteria set to 0.0001 (see Figure 3). Goal seeking convergence can sometimes be improved by reducing these values to 0.00001 or 0.000001.
 - See [Iterations panel](#) for a more in-depth discussion.
- **Changing Goal Seeking Numerical Controls** - Figure 4 shows the [Numerical Controls panel](#) in [Analysis Setup](#). The most useful parameter one can adjust is the Relative Finite Difference Step Size. Using values between 0.01 and 0.001 are recommended. In some cases using a value as high as 0.1 may be tried.

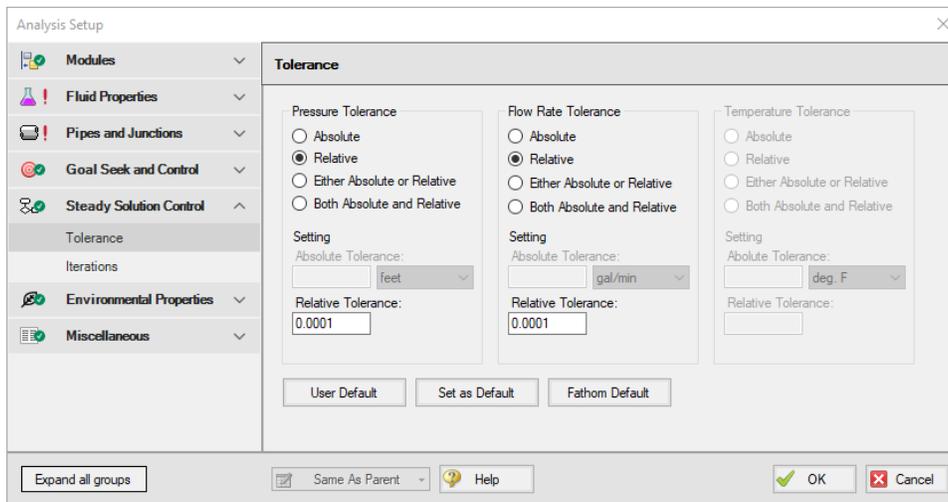


Figure 3: Reducing tolerance may resolve goal seeking difficulties

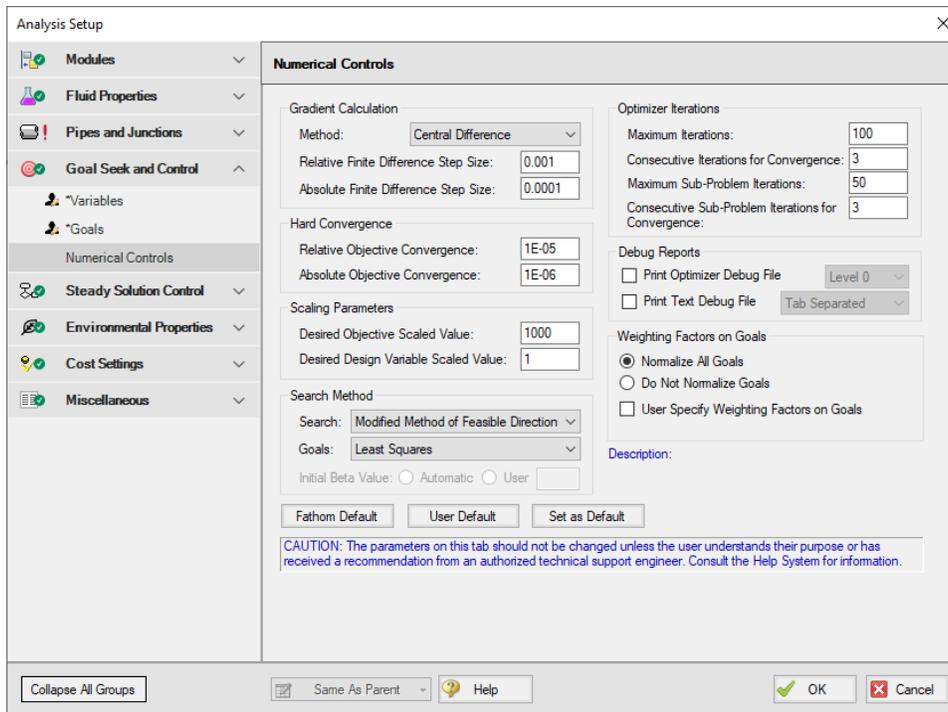


Figure 4: Changing certain numerical control parameters in the Numerical Controls panel in the Goal Seek and Control group may resolve goal seeking difficulties

Related Topics

What To Do if the Solver Gets Stuck in GSC..... 549

GSC Adjustments.....	550
GSC Inconsistencies.....	550
GSC Troubleshooting.....	545
Goals Panel.....	155
GSC Goal may not be valid. User goal = X. Final goal result = Y.....	778

What To Do if the Solver Gets Stuck in GSC

[\(GSC Module Only\)](#) Occasionally the user may have specified a goal which is in fact physically realistic, but in the process of going from a physically realistic starting point to a similarly physically realistic goal, a physically unrealistic system may be proposed by the Numerical Optimizer. What happens in such cases?

When performing goal seeking with the GSC module, AFT Fathom has been designed to gracefully handle such cases. One of two things will typically happen:

- The [Hydraulic Solver](#) will experience a computational error
- The Hydraulic Solver will not be able to converge

In the first case, AFT Fathom does not stop and tell the user of the error, but instead sets an internal flag that the system used for the particular iteration that caused the error was a poor system and should not be considered further by the Numerical Optimizer. In other words, it skips the system that caused the error. This will happen in a way which is transparent to the user and requires no intervention.

The second case is of more interest here. In this case the Hydraulic Solver gets stuck on one of the systems proposed by the Numerical Optimizer and cannot progress. The Hydraulic Solver will continue trying to converge until it reaches the iteration limit. Once it reaches that limit, it will react in a similar way to the first case above – it will set a flag and skip that system.

However, the default maximum number of iterations is 50,000 (specified on the [Iterations panel](#) in [Analysis Setup](#)). It is perfectly fine for one to wait until this limit is reached and allow the AFT Fathom to handle the situation automatically, but depending on the size of the model it may take awhile to get to 50,000 iterations. In such cases it may be desirable to allow AFT Fathom to more quickly conclude the model that will not converge by reducing the maximum iterations to something like 5,000.

Tip: If the Hydraulic Solver appears to get stuck while performing a GSC run, you can pause the Solver, open the Iterations panel, and reduce the Maximum Iterations. The default value is 50,000. Depending on the model size, a reduced value of 5,000 may be appropriate.

Related Topics

When Goals Cannot Be Achieved.....	545
GSC Adjustments.....	550
GSC Inconsistencies.....	550
GSC Troubleshooting.....	545

GSC Adjustments

([GSC Module Only](#)) When a GSC run results in the goals not being met, the user may want to adjust [GSC Numerical Controls](#) or [Solution Control](#) parameters. These can be automatically adjusted after a run if a warning has been displayed and the user is prompted to press the F2 function key. When the user does this, recommended changes are offered which the user can choose to have automatically applied. The changes will affect either the Numerical Controls panel, [Tolerance panel](#), and/or [Iterations panel](#).

Related Topics

When Goals Cannot Be Achieved.....	545
What To Do if the Solver Gets Stuck in GSC.....	549
GSC Inconsistencies.....	550
GSC Troubleshooting.....	545

GSC Inconsistencies

If an object in the [Variables panel](#) in Analysis Setup is defined with a parameter that has not been defined in that object's property window, an error will appear stating "Junction XX does not have XX parameter modeled. Cannot use as a GSC variable."

In this case, the user may resolve this inconsistency by either selecting a different parameter for the variable in the Variables panel, or by changing the specified junction to be defined using the specified variable. Note the properties window for the junction can be opened directly by double-clicking the far left column of the row on the Variables table.

Related Topics

When Goals Cannot Be Achieved.....	545
What To Do if the Solver Gets Stuck in GSC.....	549
GSC Adjustments.....	550
GSC Troubleshooting.....	545

SSL - Settling Slurry

What is the SSL Module?

An optional add-on module to AFT Fathom which allows users to simulate settling slurries.

Settling slurries are fluids which carry solids of relatively large size which cannot be suspended in the liquid under all conditions. At sufficiently low velocities the solid particles will settle to the bottom of the pipe and may ultimately plug the pipe completely. It is important to predict the behavior of such flows to avoid settling conditions and economically move the fluid and solids through the pipe.

The SSL module allows users to predict the behavior of settling slurries within the framework of the powerful AFT Fathom software. The SSL module allows users to specify solid particle properties,

distribution and concentration. State-of-the-art correlations are then used to predict the flow, pressure drop and reduced pump performance when pumping solids.

Special output reports and system curve generation are available to assist the user with understanding the hydraulic behavior of the slurry.

The SSL module can work in conjunction with the other AFT Fathom modules. The [XTS module](#) allows users to model slurry flows of constant concentration which vary over time. The [GSC module](#) allows users to vary input parameters automatically to achieve desired output parameters.

How Does the SSL Module Work?

The SSL module offers additional input windows where users can enter the properties of the solid particles. The SSL module takes the properties of the solid particles and carrier fluid (which is often water but can be another fluid) and calculates the hydraulics of the piping and pumps using widely accepted correlations.

Predicting the flow of settling slurries depends heavily on correlations based on experiment. Such predictions are inherently less accurate than standard clear fluid pipe flow.

Using the SSL Module

The user has the option of activating or not activating the SSL module when AFT Fathom first loads. After AFT Fathom is loaded, the SSL module can be activated or deactivated for use from the [Modules panel](#) in the Analysis Setup window or the Tools menu. Whether or not SSL is activated impacts the Analysis menu, Library menu and Output window as is discussed in the Modules panel topic.

If the SSL module is active, the user can still run models without slurries. This is selected under Slurry Modeling on the Analysis menu. Hence there are three possibilities for the SSL module.

- SSL is not active
- SSL is active and slurry calculation is disabled
- SSL is active and settling slurry calculation is performed

Related Topics

Modules Panel.....	85
Add-on Modules.....	538
SSL Module Fluid Properties Group.....	131
Carrier Fluid Panel.....	131
Solids Definition Panel.....	133
Slurry Definition Panel.....	136
SSL Nomenclature.....	570
SSL Engineering Assumptions.....	571

Related Examples

Beginner - Pump Sizing for Sand Transfer System - SSL	1092
Slurries with Variable Fluid Properties - SSL	1102
Slurry System Feasibility Study - SSL (Metric Only).....	1114

Slurry Calculation Method Overview

[\(SSL Module Only\)](#)

Several different slurry pipe calculation methods are offered, each with their own assumptions and requirements for input data. The type of slurry model is selected on the Slurry Definition panel.

Wilson, Addie, Clift

The Wilson, Addie, Clift slurry methodology is taken from [Wilson, et al, 2006](#). There are three Slurry Calculation Methods – Minimal, Simplified, and Detailed. These require an increasing level of knowledge and data about the slurry being modeled. Each of the methods are described with repeated information as there are many calculations that are common among the three.

- [Minimal Slurry Calculation Method](#)
- [Simplified Slurry Calculation Method](#)
- [Detailed Slurry Calculation Method](#)

4-Component

The [4-Component Slurry Model](#) is a holistic approach to characterizing how slurry flows function by considering the combined effects of four slurry flow regimes.

Wasp-Durand

The [Wasp-Durand Slurry Model](#) is a two-layer slurry approach with heterogeneous (also referred to as the Durand flow) and homogenous fluid components (also referred to as the vehicle component) typically used for coal slurry pipeline applications.

Related Topics

Sloped Pipe Theory For Slurries	568
Vertical Pipe Theory For Slurries.....	566
4-Component Slurry Model.....	558
Wasp-Durand Heterogeneous Slurry Model.....	561
SSL Engineering Assumptions.....	571

SSL Nomenclature 570

Minimal Slurry Calculation Method

([SSL Module Only](#)) The Minimal Slurry Calculation method was developed for fully-stratified flow, but can be used as a conservative method for two-layer heterogeneous flow. The following data is required for the Minimal Calculation Method:

- Temperature
- Solids concentration
- Density
- Sliding friction coefficient, μ_s

Engineering Assumptions

See the [SSL Engineering Assumptions](#) topic.

Horizontal Pipes

The pressure drop in a given by the relationship given by [Wilson, et al, 2006](#), on page 116. As described in the discussion in the book, the coefficient 0.55 is based on sand or gravel sliding in steel pipe which has a coefficient of sliding friction, μ_s of 0.40. The equation then is modified with the ratio of the value entered on the [Slurry Definition panel](#) and becomes the following:

$$\frac{i_m - i_f}{S_m - S_f} = \left(\frac{V_m}{0.55 V_{sm}} \right)^{-0.25} \left(\frac{\mu_s}{0.40} \right) \quad (1)$$

Where i_m is the dimensionless head gradient and V_m is the mean velocity based on the full diameter regardless of whether any solids collection in the bottom of the pipe has reduced the effective diameter. Note that when the Minimal input option is selected in [Analysis Setup](#) on the [Solids Definition panel](#) there is no value entered for the d_{50} or d_{85} particle sizes and therefore the M exponent, V_{50} and other parameters cannot be determined.

Related Topics

Sloped Pipe Theory For Slurries 568

Slurry Calculation Method Overview 552

Vertical Pipe Theory For Slurries 566

SSL Engineering Assumptions 571

SSL Nomenclature 570

Simplified Slurry Calculation Method

([SSL Module Only](#)) The Simplified Slurry Calculation Method was developed for two-layer heterogeneous slurry flow with a newtonian carrier fluid. This method is similar to the [Detailed Slurry Calculation Method](#), but uses a simpler method to estimate V_{50} . The following data is required for the Simplified Calculation Method:

- Temperature
- Solids concentration
- Density
- d_{50} particle size
- M exponent value

Engineering Assumptions

See the [SSL Engineering Assumptions](#) topic.

Horizontal Pipes

The correlation for additional pressure loss in horizontal pipes due to the slurry is given by [Wilson, et al. 2006](#), on page 130

$$\frac{i_m - i_f}{S_m - S_f} = 0.22 \left(\frac{V_m}{V_{50}} \right)^{-M} \quad (1)$$

Where i_m is the dimensionless head gradient and V_m is the mean velocity based on the full diameter regardless of whether any solids collection in the bottom of the pipe has reduced the effective diameter. By adding in the base fluid pressure loss, the full head gradient is given by Wilson, et al. 2006, on page 134 (note that Wilson, et al, assume the fluid is water where here the fluid is generalized)

$$i_m = \frac{f_f}{2 g D} V_m^2 + 0.22 (S_s - S_f) C_{vd} \left(\frac{V_m}{V_{50}} \right)^{-M} \quad (2)$$

Noting that:

$$(S_s - S_f) C_{vd} = S_m - S_f$$

we get the following solution for the total head gradient:

$$i_m = \frac{f_f}{2gD} V_m^2 + 0.22 (S_m - S_f) \left(\frac{V_m}{V_{50}} \right)^{-M} \quad (3)$$

Calculating the V_{50} Value

Equation 3 above needs the velocity at which 50% of the solids are suspended by the fluid, V_{50} . An estimation shown in Wilson, et al, 2006, on page 141, is used.

$$V_{50} \approx 3.93 d_{50}^{0.35} \left(\frac{S_s - 1}{1.65} \right)^{0.45} \nu_r^{-0.25} \quad (4)$$

The relative viscosity ν_r is the ratio of the actual fluid viscosity to that of water at 20 degrees C (i.e. 1.0016 centipoise).

Related Topics

Sloped Pipe Theory For Slurries.....	568
Slurry Calculation Method Overview.....	552
Vertical Pipe Theory For Slurries.....	566
SSL Engineering Assumptions.....	571
SSL Nomenclature.....	570

Related Examples

Slurries with Variable Fluid Properties - SSL	1102
---	------

Detailed Slurry Calculation Method

([SSL Module Only](#)) The Detailed Slurry Calculation Method was developed for two-layer heterogeneous slurry flow with a newtonian carrier fluid. This method is similar to the [Simplified Slurry Calculation Method](#), but includes more sophisticated methods to calculate M and V_{50} , as is discussed below. The following data is required for the Detailed Calculation Method:

- Temperature
- Solids concentration
- Density
- d_{50} particle size
- d_{85} particle size if the M exponent is to be calculated

- M exponent value if no d_{85} particle size is available
- Terminal velocity parameter, using either the Shape Factor (K) or V_t / V_{ts}

Engineering Assumptions

See the [SSL Engineering Assumptions](#) topic.

Horizontal Pipes

The correlation for additional pressure loss in horizontal pipes due to the slurry is given by [Wilson, et al, 2006](#), on page 130

$$\frac{i_m - i_f}{S_m - S_f} = 0.22 \left(\frac{V_m}{V_{50}} \right)^{-M} \quad (1)$$

Where i_m is the dimensionless head gradient and V_m is the mean velocity based on the full diameter regardless of whether any solids collection in the bottom of the pipe has reduced the effective diameter. By adding in the base fluid pressure loss the full head gradient is given by Wilson, et al, 2006, on page 134 (note that Wilson, et al, assume the fluid is water where here the fluid is generalized)

$$i_m = \frac{f_f}{2gD} V_m^2 + 0.22 (S_s - S_f) C_{vd} \left(\frac{V_m}{V_{50}} \right)^{-M} \quad (2)$$

Noting that:

$$(S_s - S_f) C_{vd} = S_m - S_f$$

We get the following solution for the total head gradient:

$$i_m = \frac{f_f}{2gD} V_m^2 + 0.22 (S_m - S_f) \left(\frac{V_m}{V_{50}} \right)^{-M} \quad (3)$$

Calculating the V_{50} Value

Equations 2 and 3 above need the velocity at which 50% of the solids are suspended by the fluid, V_{50} . This is discussed by Wilson, et al, 2006, with a functional relationship given on page 130.

$$V_{50} = w_{50} \sqrt{\frac{8}{f_f}} \cosh(60 d_{50}/D) \quad (4)$$

Where:

- w_{50} is the particle associated velocity
- d_{50} is the particle diameter
- D is the pipe inner diameter

w_{50} used in Equation 4 above, is the particle associated velocity for the median particle size, and is defined in equation 5 below. The 3rd edition of Wilson (Wilson, et al, 2006, from which the equations for the rest of this discussion are taken) defines w using Figure 6.5 on page 136 of the text. The plot of w vs. particle size is based on an empirically-derived equation in the 2nd edition of the text (Wilson, et al, 1996), equation 6.12 found on page 134. The SSL module uses that equation (Equation 5 below) in its solution methodology.

$$w_{50} = 0.9v_t + 2.7 \left[\frac{(\rho_s - \rho_f) g \mu}{\rho_f^2} \right]^{1/3} \quad (5)$$

Where:

- v_t is the particle settling velocity for the particle size in question
- ρ_s is the slurry density
- ρ_f is the fluid density

Calculating the M Exponent Value

If both the d_{50} and d_{85} values are entered, the M exponent value can be calculated using the method shown in Wilson, et al, 2006, on page 140.

$$M = (0.25 + 13 \sigma_s^2)^{-0.50} \quad (6)$$

Where:

$$\sigma_s = \log \left[\frac{w_{85} \cosh(60 d_{85}/D)}{w_{50} \cosh(60 d_{50}/D)} \right] \quad (7)$$

When using these calculations M is limited to between 0.25 and 1.7.

Related Topics

Sloped Pipe Theory For Slurries.....	568
Slurry Calculation Method Overview.....	552
Vertical Pipe Theory For Slurries.....	566
SSL Engineering Assumptions.....	571
SSL Nomenclature.....	570

Related Examples

Beginner - Pump Sizing for Sand Transfer System - SSL.....	1092
Slurry System Feasibility Study - SSL (Metric Only).....	1114

4-Component Slurry Model

([SSL Module Only](#)) The 4-Component Slurry Model is a holistic approach to characterizing how slurry flows function by considering the combined effects of four slurry flow regimes. The model strongly references and utilizes the method of the same name outlined by [GIW Industries Inc.](#)

The 4-Component Model can be selected on the Slurry Model dropdown in the [Slurry Definition panel](#) of Analysis Setup. Optionally, a [Solids Library](#) can be created to define the slurry properties.

Engineering Assumptions

See the [SSL Engineering Assumptions](#) topic.

The Four Slurry Regimes

The four slurry flow regimes are:

- Carrier Fluid
- Pseudo-homogeneous
- Heterogeneous
- Fully Stratified

The Carrier Fluid regime considers particles smaller than 40 μm . These solid particles mix with liquid in such a way that mixture viscosity is affected.

The Pseudo-homogeneous regime considers particles larger than 40 μm and smaller than 200 μm . Due to the turbulence, the solid particles are supported within the fluid. The mixture's viscosity is not affected by this regime.

The Heterogeneous regime considers particles larger than 200 μm and smaller than 1.5% of the pipe's diameter (μm). Particles of this size begin to settle creating a heterogeneous mixture where the solids are found in greater concentration at the bottom of the pipe. Turbulence in the fluid and contact with the pipe wall support these solids from settling out of the fluid.

The Fully Stratified regime considers particles larger than 1.5% of the pipe's diameter (μm). Solids are primarily supported by the pipe wall and will slide along the bottom of the pipe. A sliding bed friction model is used for this flow regime.

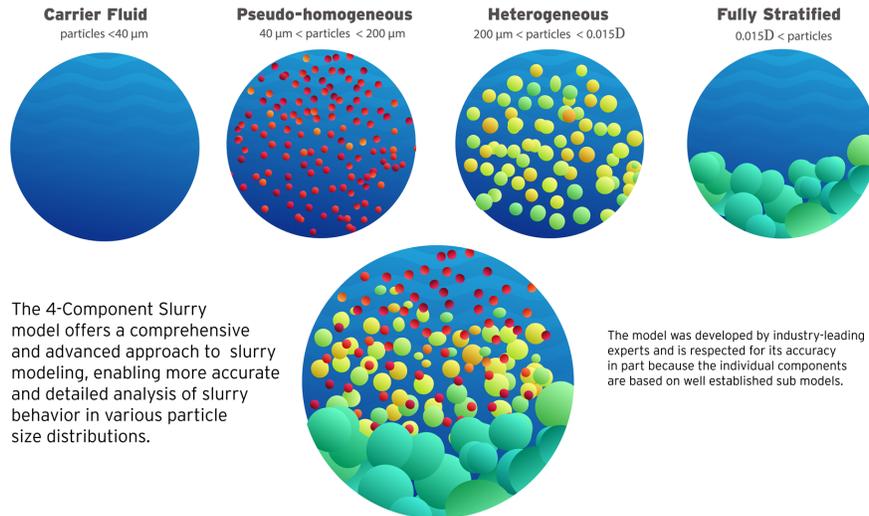


Figure 1: The 4-Component Slurry Model considers combined effects from four slurry regimes

The water pressure gradient i_m (in meters of water head per meter of pipe) from frictional losses is the sum of i_f , the pressure gradient of the carrier fluid component, with the excess pressure gradients of the pseudo-homogeneous component Δi_p , the heterogeneous component Δi_h , and the fully stratified component Δi_s . The slurry pressure gradient j_m is related to i_m by the specific gravity of the total slurry mixture S_m .

$$i_m = i_f + \Delta i_p + \Delta i_h + \Delta i_s$$

$$j_m = \frac{i_m}{S_m}$$

Defining and Using a 4-Component Slurry Model

To begin defining and using the 4-Component Slurry Model, enable the [Settling Slurry Module](#) with the radio button set to Settling. Define the Carrier Fluid on the [Carrier Fluid panel](#). On the Slurry Definition panel, set the Slurry Model dropdown to Settling Slurry - 4-Component Model. Navigate to Solids Definition panel and select one of the Solids Specifications options. If a Solids library is attached, it can likewise be selected from the dropdown. Density, d100 (the Maximum Solid Diameter), and a Particle Size Distribution are required to be filled out. Once the Solids Definition panel is completely defined, navigate to the Slurry Definition panel. Enter an Amount Solids Added and Sliding Friction Coefficient in the required fields. With all of the required information entered, the 4-Component Slurry Model can now be used.

4-Component [SSL Pump De-Rating](#) is available for use with 4-Component slurries.

Related Topics

Sloped Pipe Theory For Slurries.....	568
Slurry Calculation Method Overview.....	552
Vertical Pipe Theory For Slurries.....	566
Wasp-Durand Heterogeneous Slurry Model.....	561
SSL Engineering Assumptions.....	571
SSL Nomenclature.....	570

Wasp-Durand Heterogeneous Slurry Model

([SSL Module Only](#)) The Wasp-Durand Slurry Model is a two-layer slurry approach with heterogeneous (also referred to as the Durand flow) and homogenous fluid components (also referred to as the vehicle component). This method started with the heterogeneous [Durand and Condolios](#) model in the 1950s, was expanded by [Wasp et al.](#) in the 1960s and 1970s, and has continued to evolve based on the research of [Liu Dezhong](#), [Fei Xiangjun](#), [Richard W. Hanks](#) and others. This method is used most often with long distance coal pipelines, particularly for coal slurries and slurries with relatively fine particles.

The homogenous component incorporates the viscosity effects of fine particles, similar to the [4-Component Slurry Model](#), but there is no hard size boundary on which particles can affect the viscosity but finer particles have a greater impact. Similarly, particles of any size may also contribute to the heterogeneous component depending on the degree of stratification for their size group, making Wasp-Durand a model well-suited for finer slurries that may encounter modeling limitations when using Wilson, Addie, Clift or 4-Component slurry models.

Engineering Assumptions

See the [SSL Engineering Assumptions](#) topic.

Definition of Solids

Use of Wasp-Durand requires a Granulometric Spread Distribution. This particle size distribution may be defined in either the [Solids Library](#) or the [Solids Definition Panel](#) of the Analysis Setup Window. A Granulometric Spread Distribution is a histogram-like distribution defined in terms of the fraction of total solids in each size group and the mean particle diameter of each size group.

Stratification in Wasp-Durand is calculated per size group as a ratio of the concentration of solids near the top of the pipe to the concentration of solids towards the middle of the pipe, as shown in Figure 1.

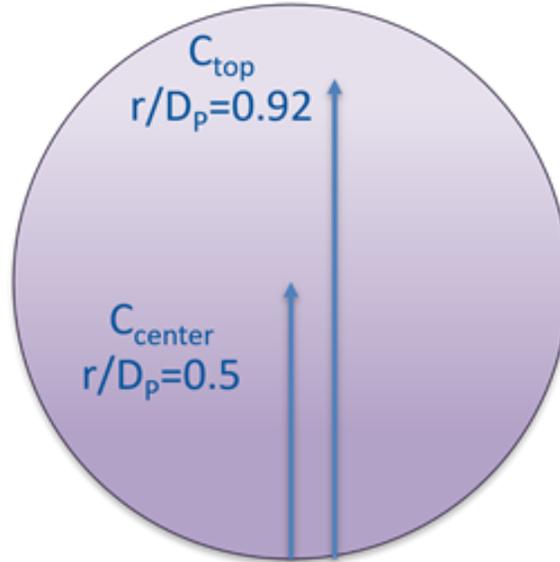


Figure 1: A cross-sectional illustration of the locations within a pipe where Wasp-Durand top and center solids concentrations are defined

Different industries and authors may use different notations for the Wasp-Durand stratification term, but any of the forms shown in Equation 1 are equivalent.

$$\frac{C_v}{C_{v,A}} = \frac{C_{top}}{C_{center}} = \frac{C}{C_A} \quad (1)$$

Wasp-Durand is essentially a modular approach, and can be customized using the [Wasp-Durand Options area of the Slurry Definition panel](#).

Deposition Velocity

The deposition velocity is the velocity where solids of a given size have come out of the slurry forming a stationary bed on the bottom of the pipe. For Wasp-Durand, the selection of heterogeneous calculation method determines which method is used for the deposition velocity calculation. The Liu Dezhong deposition velocity calculation is shown in Equations 2 and 3.

$$V_{sm} = 9.5 \left[gD \left(\frac{\rho_s}{\rho_h} - 1 \right) V_t \right]^{1/3} C_V^{1/m} \left(\frac{V_{tw}}{V_t} \right)^{1/6} \quad (2)$$

$$m = \begin{cases} 4.2857 \frac{C_{top}}{C_{center}} + 2.5714 & \text{if } \frac{C_{top}}{C_{center}} < 0.8 \\ 6 & \text{if } \frac{C_{top}}{C_{center}} \geq 0.8 \end{cases} \quad (3)$$

Where ρ_s is the solid density, ρ_h is the homogenous component density, C_V is the volumetric concentration of solids, V_t is the weighted average of sedimentation/terminal velocities of each particle size group in the homogeneous/vehicle fluid, and V_{tw} is the weighted average of particle sedimentation velocities in a pure carrier fluid (i.e. neglecting viscosity and density effect of fine solids). The m exponent is constant for quasi-homogeneous flow, and variable for more heterogeneous flow regimes.

This version of the Liu Dezhong deposition velocity calculation with the m exponent is from the Chinese national slurry transport code, and differs from the original form in which m is always 6.

Note: The sedimentation velocity terms are calculated using whichever method is specified as the Sedimentation Velocity Model.

The Wasp and Fei Xiangjun equations for deposition velocity are shown in Equations 4 and 5, respectively.

$$V_{sm} = 3.113 C_V^{0.1858} \left[2gD \left(\frac{\rho_s - \rho_m}{\rho_m} \right) \right]^{1/2} \left(\frac{d_{85}}{D} \right)^{1/6} \quad (4)$$

$$V_{sm} = \frac{2.26}{\sqrt{f_h}} \left[gD \left(\frac{\rho_s - \rho_m}{\rho_m} \right) \right]^{0.5} C_V^{1/4} \left(\frac{d_{90}}{D} \right)^{1/3} \quad (5)$$

Where ρ_m is the overall slurry density and f_h is the homogeneous Darcy friction factor. Different particle diameters are used by different sources for the Wasp deposition velocity calculation, and we follow Liu Dezhong's convention of using d_{85} .

Sedimentation Velocity

The sedimentation velocity is per particle size group and is the terminal particle free-fall velocity that occurs when the drag, gravity, and buoyancy forces on a particle are balanced. Both the Ruby & Zanke and Liu Dezhong approaches can be used across a wide range of particle sizes and flow conditions, including laminar, transition, and turbulent flow. Equations 6 and 7 show the Ruby & Zanke (1977) approach for terminal settling velocity.

$$V_{t,i} = \frac{10 v_h}{d_i} \left[\sqrt{1 + \frac{\rho_s - \rho_h}{\rho_h} \frac{g d_i^3}{100 v_h^2}} - 1 \right] \quad (6)$$

$$v_h = \frac{\mu_h}{\rho_h} \quad (7)$$

Where $V_{t,i}$ is the terminal settling velocity for each particle size group i and μ_h is the dynamic viscosity of the homogeneous component.

The Liu Dezhong sedimentation velocity calculation is shown in Equations 8-11. The particle size number for each particle size group $N_{d,i}$ and μ_f is the clear fluid viscosity. The homogeneous Bingham stiffness factor μ_{ph} is calculated as shown in Equations 22 and 23.

$$V_{t,i} = \left[g \left(\frac{\rho_s}{\rho_h} - 1 \right) \frac{\mu_{ph}}{\rho_h} \right]^{1/3} N_{\omega,i} \quad (8)$$

$$N_{\omega,i} = \frac{20.5209}{N_{d,i}} \left[\left(1 + 0.105587812 N_{d,i}^{1.5} \right)^{0.5} - 1 \right]^2 \quad (9)$$

The weighted average sedimentation velocity is shown in Equation 10. The decimal solid fraction in each band from the Granulometric Spread Distribution is w_i .

$$V_t = \sum w_i V_{t,i} \quad (10)$$

Heterogeneous Gradient

The total slurry gradient takes the form shown in Equation 11, where i_1 is the homogenous gradient and Δi_2 is the heterogeneous contribution. The homogeneous gradient will depend on which slurry viscosity model is applied and whether or not Hanks' Bingham plastic is utilized, while the heterogeneous component is calculated with the method specified by the user in the Slurry Definition panel.

$$i = i_1 + \Delta i_2 \quad (11)$$

The Wasp-Durand heterogeneous gradient is shown in Equations 12-14, where C_{1V} is the volumetric concentration of the homogeneous portion, C_{2V} is the volumetric concentration of the heterogeneous portion, and C_{1D} is the settlement resistance coefficient of particles in the homogenous fluid.

$$\Delta i_2 = K i_1 \left[\frac{gD \left(\frac{\rho_s}{\rho_h} - 1 \right)}{v^2} \right]^{1.5} \sum (C_{2V})_i (C_{1D})_i^{-0.75} \quad (12)$$

$$(C_{1D})_i = \frac{4gd_i \left(\frac{\rho_s}{\rho_h} - 1 \right)}{3V_{t,i}^2} \quad (13)$$

$$K = \begin{cases} 150 & \text{if } C_{1V} > C_{2V} \\ 82 & \text{if } C_{1V} < C_{2V} \end{cases} \quad (14)$$

The heterogeneous gradient method for Liu Dezhong is defined in Equations 15 and 16, while Equation 17 defines the Fei Xiangjun approach.

$$\Delta i_2 = \frac{k}{V} \left[\frac{\rho_s}{\rho_h} - 1 \right] \sum (C_{2V})_i V_{t,i} \quad (15)$$

$$k = 15 \frac{C_{top}}{C_{center}} + 1.2 \quad (16)$$

$$\Delta i_2 = \frac{4.07 V_t C_V}{V} \left[\frac{\rho_s}{\rho_h} - 1 \right] \quad (17)$$

Apparent Viscosity

The option to Apply Apparent Slurry Properties allows the viscosity of the homogenous portion to be impacted by the presence of solid particles. These can be Newtonian viscosities, or a non-Newtonian approximation if the Hanks' Bingham Plastic option is enabled. This is described in the next section.

The Newtonian Fei Xiangjun model for apparent viscosity is defined in Equation 18, where C_{Vm} is the maximum volumetric concentration.

$$\mu_h = \mu_f \left(1 - \frac{C_V}{C_{Vm}} \right)^{-2} \quad (18)$$

The Thomas (1965) model for apparent dynamic viscosity of the homogenous component is shown in Equation 19.

$$\mu_h = 1 + 2.5C_V + 10.05C_V^2 + 0.00273e^{16.6C_V} \quad (19)$$

Hanks' Bingham Plastic

The Hanks' Bingham Plastic Approach was developed by Richard W. Hanks. This is a two-layer model that assumes slug or plug flow in an annular distribution, with a constant velocity slug surrounded by a non-Newtonian boundary layer in which frictional losses are concentrated. The general expression for [Bingham Plastic](#) shear stress is shown in Equation 20, where τ_0 is the yield stress and μ_{ph} is the effective Bingham stiffness factor of the Wasp-Durand vehicle component.

$$\tau = \tau_0 + \mu_{ph} \frac{du}{dy} \quad (20)$$

In the context of the Wasp-Durand approach, Hanks' Bingham Plastic is used to calculate the friction from the vehicle opponent and allows treating this component as apparently non-Newtonian. The yield stress may be user-specified, or calculated using the Fei Xiangjun method in Equation 21.

$$\tau_0 = 0.098 \exp \left(B \frac{C_V - C_{V0}}{C_{Vm}} + 1.5 \right) \quad (21)$$

Where C_{V0} is the boundary concentration between the Newtonian and non-Newtonian viscosity regimes and C_{Vm} is the maximum volumetric concentration. B is an empirical coefficient equal to 8.45 for mineral slurries and 6.87 for coal slurries. The yield stress is a fluid property, and may be calculated in the Analysis Setup Window without running the solver.

Fei Xiangjun also defines a method for calculating the Bingham stiffness factor, shown in Equations 22 and 23, which is used when Fei Xiangjun is selected as the apparent viscosity model and Hanks' Bingham Plastic is enabled in Wasp-Durand options.

$$\mu_{ph} = \mu_f \left(1 - K \frac{C_V}{C_{Vm}} \right)^{-2.5} \quad (22)$$

$$K = 1 + 2 \left(\frac{C_V}{C_{Vm}} \right)^{0.3} \left(1 - \frac{C_V}{C_{Vm}} \right)^4 \quad (23)$$

The clear fluid friction factor is μ_f .

Related Topics

Sloped Pipe Theory For Slurries.....	568
Slurry Calculation Method Overview.....	552
Vertical Pipe Theory For Slurries.....	566
4-Component Slurry Model.....	558
SSL Engineering Assumptions.....	571
SSL Nomenclature.....	570

Vertical Pipe Theory For Slurries

([SSL Module Only](#)) Flow in vertical pipes offer a special-case flow regime. The basic equation for flow rising in a vertical pipe is given in [Wilson, et al, 2006](#), on page 178.

$$\frac{dP}{dz} = \rho_w g [1 + (S_s - 1) C_{vi}] + \frac{\rho g f V_m^2}{2 g D} \quad (1)$$

It is important to note that this is the total pressure drop per unit length whereas the [horizontal pipe equations](#) have components for the pure fluid and the solids separately.

Noting that for water, for which Equation 1 is based, we can substitute:

$$S_m = 1 + (S_s - 1) C_v$$

and obtain the following basic relationship:

$$\frac{dP}{dz} = \rho_w g S_m + \frac{\rho g f V_m^2}{2 g D} \quad (2)$$

Wilson, et al, states that for particles smaller than 150 microns, the density in the final term should be the mixture density:

$$\rho_m = \rho_w S_m$$

Substituting this in to Equation 2 we obtain:

$$\frac{dP}{dz} = \rho_w g S_m \left(1 + \frac{f_w V_m^2}{2 g D} \right) \quad (3)$$

The friction factor f could be found with homogenous methods, but it is often reasonable to use the friction factor for a water flow f_w with the same mean velocity. For larger particles Wilson, et al, suggests that the density term should be that of water which, when substituted in to Equation 2 yields:

$$\frac{dP}{dz} = \rho_w g \left(S_m + \frac{f_w V_m^2}{2 g D} \right) \quad (4)$$

Related Topics

Sloped Pipe Theory For Slurries.....	568
Slurry Calculation Method Overview.....	552
4-Component Slurry Model.....	558
Wasp-Durand Heterogeneous Slurry Model.....	561
SSL Engineering Assumptions.....	571
SSL Nomenclature.....	570

Sloped Pipe Theory For Slurries

(SSL Module Only)

Wilson, Addie, Clift Settling Slurry

Sloped pipe calculations are a combination of [horizontal head loss](#) and a [vertical component](#). A widely used formula is given in [Wilson, et al, 2006](#), on page 183. This is used for all [Wilson, Addie, Clift](#) slurry models.

$$\Delta i(\theta) = \Delta i(0) \cos \theta + (S_s - 1)C_{vd} \sin \theta \quad (1)$$

Where $\Delta i(\theta)$ is simply the excess pressure gradient ($i_m - i_w$) for a pipe at an angle θ .

The term $\Delta i(\theta)$ is the solids effect for a horizontal pipe and is calculated using the equations described in that section above. It is modified by $\cos(\theta)$ to reflect the pipe inclination from horizontal.

The Deposition Velocity, V_{sm} , is also modified based on pipe angle. The following is added to the value calculated for horizontal pipes.

$$\Delta_D \sqrt{2gD(S_s - 1)} \quad (2)$$

Where Δ_D is determined from the graph in Wilson, et al, 2006, on page 181.

4-Component Settling Slurry

The sloped pipe corrections used for the [4-Component Slurry Model](#) developed by [GIW Industries Inc](#) are similar to the Wilson, Addie, Clift approaches shown above, but have been adapted for use with multiple components and the graphical method for Δ_D has been replaced with an analytical expression. Settling velocity corrections are shown in Equations 3 and 4.

$$V_{sm,\theta} = V_{sm,horizontal} + \Delta_D \sqrt{2gD(S_s/S_f - 1)} \quad (3)$$

$$\Delta_D = 0.75 \theta - \frac{0.50 (0.6366 \theta)^2}{1 - 0.6366 \theta} \quad \text{for } \theta > 0 \quad (4)$$

$$\Delta_D = 0.75 \theta - \frac{0.02 (2.29 \theta)^2}{1 - 2.29 \theta} \quad \text{for } \theta < 0$$

Where D is the pipe diameter in meters, θ is the pipe angle in radians, S_s is the solid specific gravity and S_f is the specific gravity of the carrier portion of the fluid which is affected by particles smaller than 40 microns. A plot of Δ_D is shown in Figure 1.

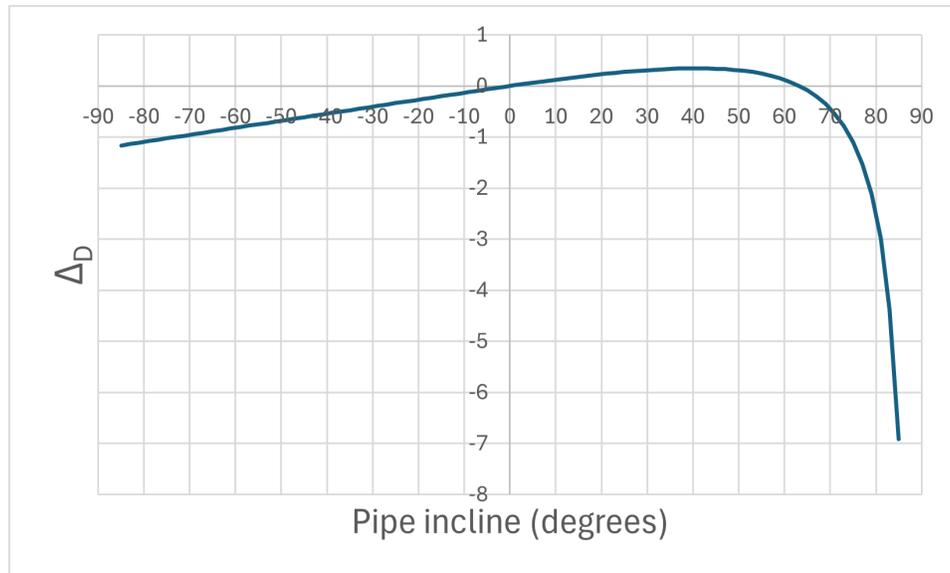


Figure 1: A plot of the analytical form of delta D used for 4-Component slope corrections

The corrected pressure gradient is shown in Equation 5. Note that the solutions to the heterogeneous gradient component Δi_h and the stratified gradient component Δi_s are affected by the change in deposition velocity, and therefore differ from their horizontal values.

$$i_m = i_f + \Delta i_p + (\Delta i_h + \Delta i_s) \cos \theta \tag{5}$$

Wasp-Durand Heterogeneous Model

The [Wasp-Durand Heterogeneous Slurry Model](#) does not account for inclined pipes or vertical conveying effects. For this reason, a [caution message](#) will be displayed for pipes with an angle greater than 8 degrees, and a [warning message](#) will be shown for pipes angled beyond 18 degrees.

Related Topics

Slurry Calculation Method Overview.....	552
Vertical Pipe Theory For Slurries.....	566
4-Component Slurry Model.....	558
Wasp-Durand Heterogeneous Slurry Model.....	561
SSL Engineering Assumptions.....	571
SSL Nomenclature.....	570

SSL Nomenclature

(SSL Module Only)

C_E	derated pump multiplier on efficiency
C_H	derated pump multiplier on head
C_m	mass concentration of solids
C_{vd}	volume concentration of solids, delivered
C_{vi}	volume concentration of solids, in situ
C_{vb}	volume concentration of solids in sliding bed (bed concentration)
d_{50}	mass-median particle diameter
d_{85}	diameter for which 85% by mass of the particles are finer
d_{100}	max particle diameter size (100% by mass of the particles are finer)
D	inner diameter of pipe
f	friction factor (Darcy)
f_f	friction factor when calculating i_{fg}
f_{turb}	turbulent friction factor
f_w	friction factor of water
g	acceleration due to gravity
i	head gradient
i_f	head gradient contribution from fluid
i_m	value of i for flow in terms of water density
i_s	head gradient contribution from solids
i_w	head gradient contribution from fluid when fluid is water
j_m	head gradient in terms of slurry density
K	loss factor
K	Power Law constant
L	pipe length
L_{eq}	equivalent pipe length of fittings
M	an exponent in the stratification ratio equation
ρ_w	density of water

S_s	relative density of solids in slurry
S_f	relative density of fluids in slurry
S_m	relative density of slurry
μ_s	coefficient of sliding bed friction
V_{50}	value of V_m at which 50% of the solids are suspended in the fluid
V_m	mean velocity of mixture
V_{sm}	deposition velocity, velocity at which particles have all settled out
V_t	terminal velocity of a solid particle in the slurry carrier fluid
V_{ts}	terminal velocity of a perfectly spherical solid particle in the slurry carrier fluid

SSL Engineering Assumptions

[\(SSL Module Only\)](#)

Calculations done with the SSL module are based on the following assumptions:

Applicable to All Slurry Methods

- Slurry models are empirical models developed from steady state flow data.
- Solid particles are moving with the fluid. Stationary beds cannot be modeled. Sliding beds can be modeled with caution.
- Dynamic mixing of slurries with different solid concentrations is not modeled.
- Heat transfer is not modeled.
- The settling of slurries is modeled in pipes only, and settling is not modeled within junctions or components.

Wilson, Addie, Clift Slurry Model

- Base liquid behaves in a Newtonian viscosity manner.
- Slurry viscosity is equivalent to the viscosity of the carrier fluid.
- Particles below 150 μm are treated as part of the carrier fluid.
- [Minimal Slurry Calculation Method](#) assumes all particles are in a sliding bed and none are suspended by turbulence.
- [Simplified](#) and [Detailed](#) methods assume a sliding friction coefficient of 0.45.
- The velocity used to calculate the friction factor in the deposition velocity limiting equation is approximated as the current pipe velocity.

4-Component Slurry Model

- Carrier fluid component behaves in a Newtonian viscosity manner.
- Only particles below 40 µm affect carrier fluid viscosity.
- The deposition velocity nomograph is assumed to follow the same behavior as the Wilson, Addie, Clift nomograph.
- The limiting equation for deposition velocity is not used.

Wasp-Durand Slurry Model

- Carrier fluid viscosity is assumed to be Newtonian unless the Hank's Bingham Plastic option is used.
- Particle distribution along the vertical axis of the pipe cross-section is assumed to be continuous. Due to this, the heterogeneous gradient will be underestimated in systems with significant sliding beds.
- All pipes are assumed to be horizontal.
- All particle size groups are assumed to contribute to both the homogeneous and heterogeneous flow components with larger particles contributing more to the heterogeneous (Durand) portion and smaller particles contributing more to the homogeneous (vehicle) portion.

SSL Flow Rates

([SSL Module Only](#)) When specifying or referring to slurry flow rate, three different meanings of flow rate are possible and two are commonly used. The definitions for flow rate are as follows:

- **Slurry** - Combined flow rate of the fluid and solids. This is the most common flow rate definition.
- **Solids** - Flow rate of just the solids in the slurry. This is often used because the purpose of the slurry is frequently to convey solids and this flowrate is of the greatest importance to the engineer.
- **Clear Liquid** - Flow rate of the carrier fluid. This is not commonly used.

The three flow rate definitions are of course interrelated. The slurry flow rate is the sum of the other two. As shown in the [slurry output](#) section, all three flow rates can be displayed in the Output window.

The three flow rate definitions can be specified in three places in AFT Fathom. Two are on the [Pump Properties Window](#) and one is on the [Assigned Flow Properties](#) window.

An example of one of the three locations is shown in Figure 1 in the Pump Properties Window. When the SSL module is in use there is a dropdown list adjacent to the flowrate specification fields (as shown in Figure 1) that allows one to choose one of the three. The default selection is the flow rate of slurry.

Note that anywhere in AFT Fathom that specifies or displays flow rate information and does not offer the qualifier of slurry/solids/clear liquid, the flow rate of slurry is assumed.

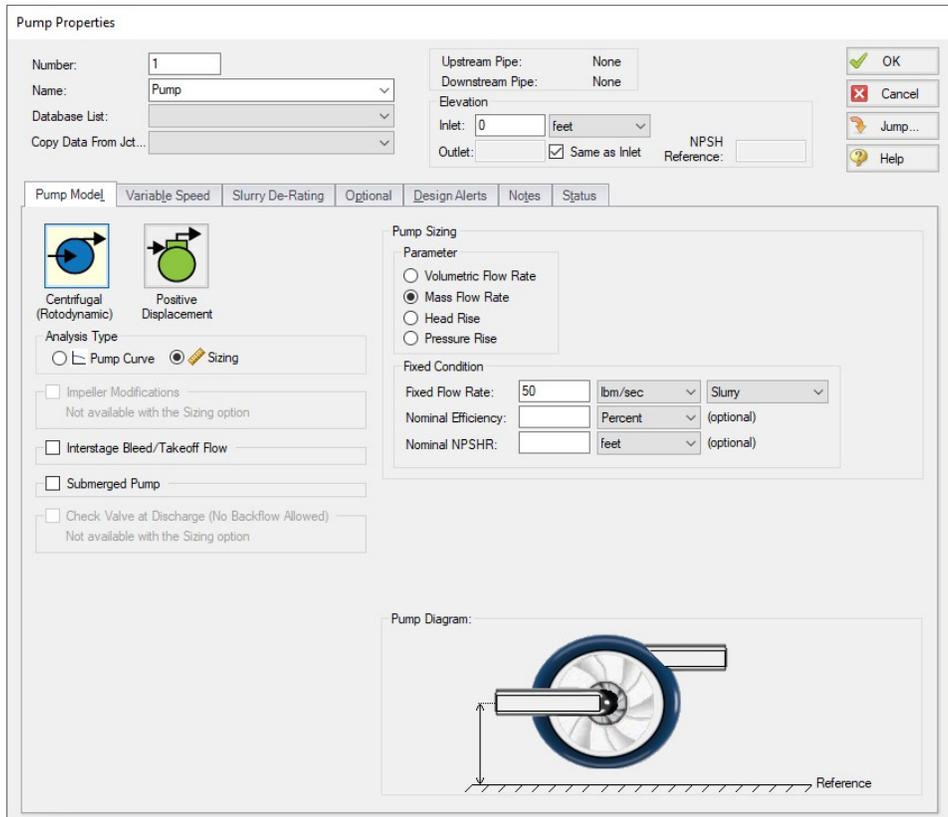


Figure 1: Pump Properties Window with fixed mass flow rate of slurry specified

Related Topics

SSL Engineering Assumptions.....	571
SSL Nomenclature.....	570

Estimating M From Recorded Data

(SSL Module Only) There is an option on the [Solids Definition panel](#) of the [Analysis Setup window](#) and [Solids Library](#) to estimate M from recorded data. Choose the button for M Curve Fit. This is based on discussion in [Wilson, et al, 2006](#), on pages 128-136.

Figure 1 shows the M Parameter Curve Fit window. Data for velocity and relative excess pressure gradient are entered and a linear curve fit is generated on a log-log plot. The M factor is displayed at the top. Data that does not fit the curve can be disregarded by the user by deselecting the checkbox in the table.

M values are not allowed to be greater than 1.7 or less than 0.25 based on recommendations in [Wilson, et al.](#)

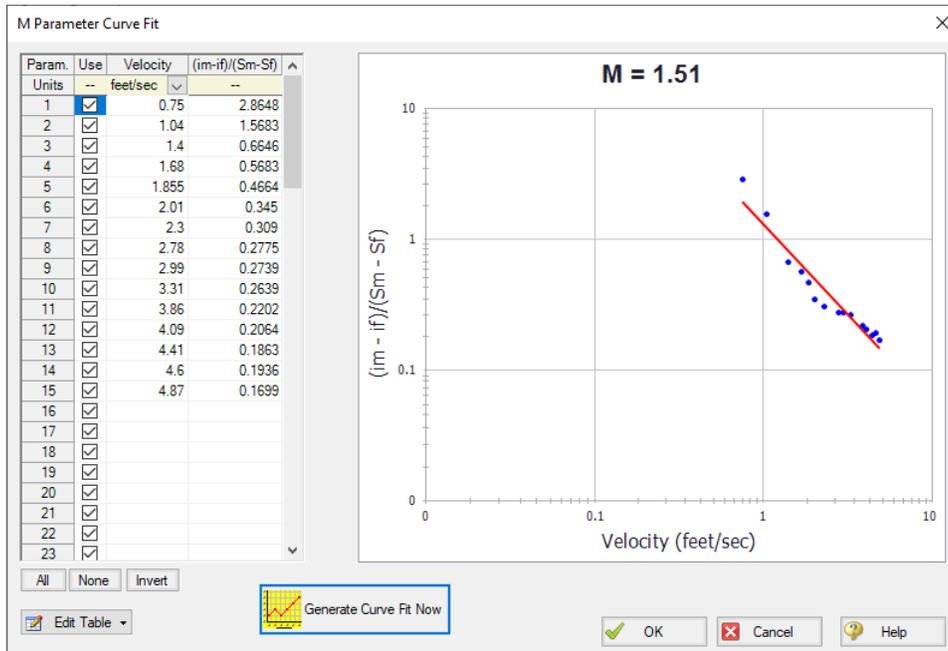


Figure 1: M curve fit window based on recorded data

Related Topics

SSL Engineering Assumptions.....	571
SSL Nomenclature.....	570

Related Examples

Beginner - Pump Sizing for Sand Transfer System - SSL.....	1092
--	------

Solids Library

(SSL Module Only) Users can create Solids Library entries which retain data for specific solids profiles. The Solids Library window (Figure 1) can be accessed from the Library menu or from [Analysis Setup](#) on the [Solids Definition panel](#).

This allows you to view, add, edit and delete solids in the library. The physical properties are entered in the Change Solids Physical Properties window. Enter the name of the solids data and the temperature units.

Often the physical properties of the solids do not vary appreciably with temperature or the variation is unknown. In this case select Use Constant Properties (Figure 2). The temperature units are ignored but one must still be selected. The density is always required. The specific heat and thermal conductivity are optional and are only used when heat transfer is modeled.

Cost Calculation

If one or more properties vary with temperature, uncheck the Use Constant Properties box. The property values as a function of temperature can then be entered in the table on the Solids Property Data tab (Figure 2). If a property does not vary with temperature, simply enter the same value for each temperature listed in the table.

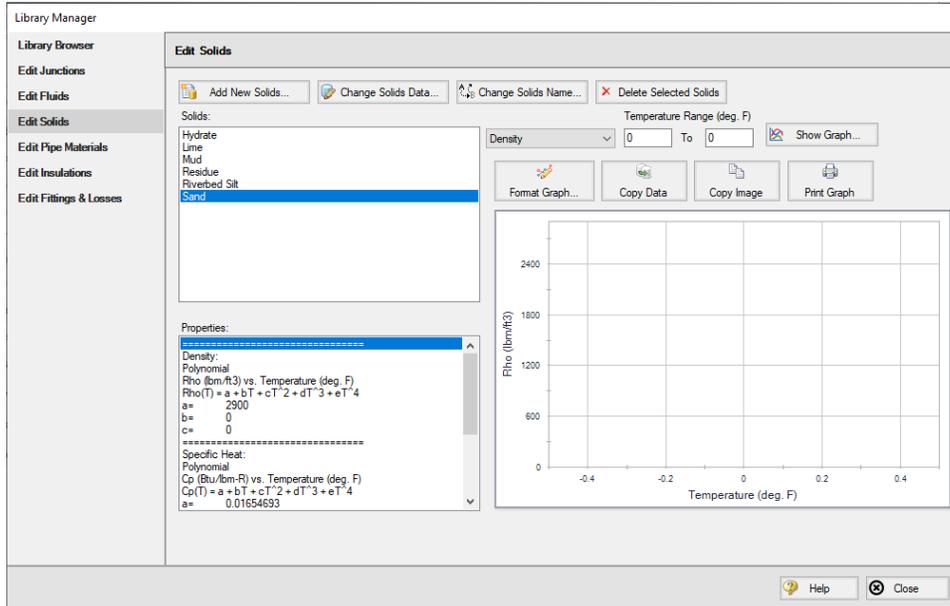


Figure 1: Solids Library showing example of user-entered solids data

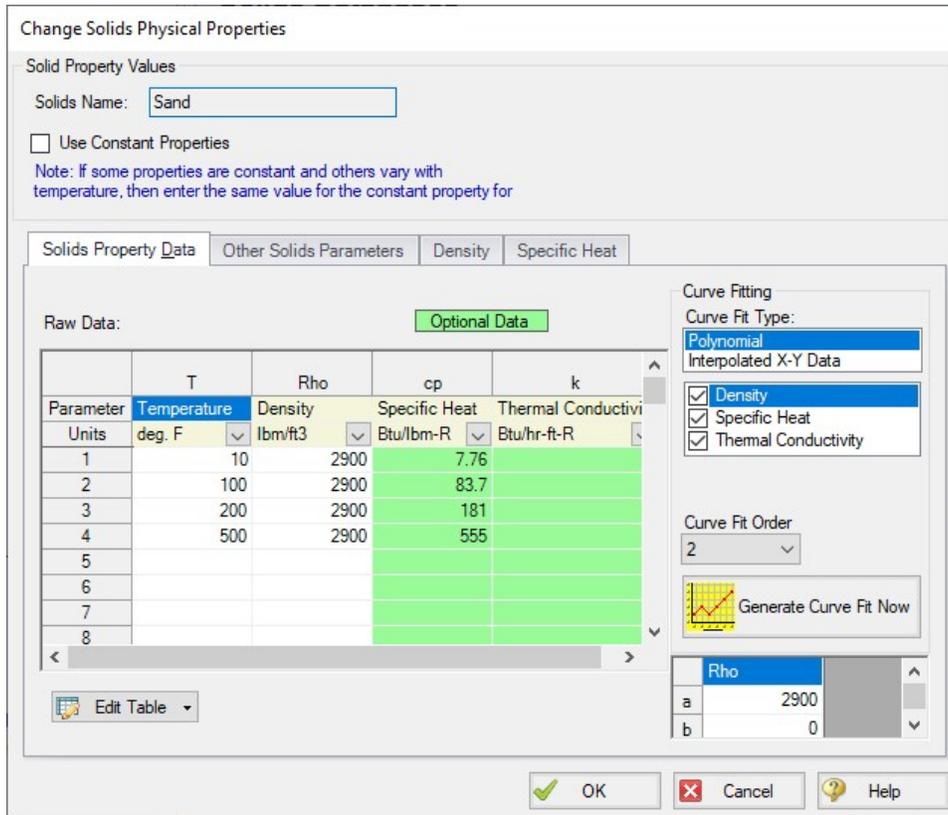


Figure 2: Change Solids Physical Properties window showing tab to specify thermophysical properties of solids (available when “Use Constant Properties” is unchecked)

Other Solids Parameters

The Terminal Velocity Parameter and the M exponent are entered on the Other Solids Parameters tab (Figure 3). The particle size distribution is also entered on this tab.

Similar to direct entry on the [Solids Definition panel](#), the user can allow AFT Fathom to [estimate the \$M\$ factor from recorded data](#) or can input an M value directly. If M is calculated, values for both d_{50} and d_{85} must be entered.

The Terminal Velocity Parameter can be entered as either a Shape Factor (K) or the ratio of the particle's terminal velocity (V_t) to that of a sphere (V_{ts}). This ratio is designated by z or X_i

Typical shape factors are given in [Wilson, et al, 2006](#), on page 46 where $K = 0.26$ for sand. To ignore this effect select $V_t / V_{ts} = 1$.

Particle Size Distribution

The particle size parameters d_{50} and d_{85} can be entered directly or statistically calculated based on particle size distribution data.

With the Use Particle Size Distribution option checked, you can enter the percent of the solids that remained on the mesh as observed from test data for each Tyler Mesh Size. The d_{50} and d_{85} parameters will automatically be calculated (Figure 3).

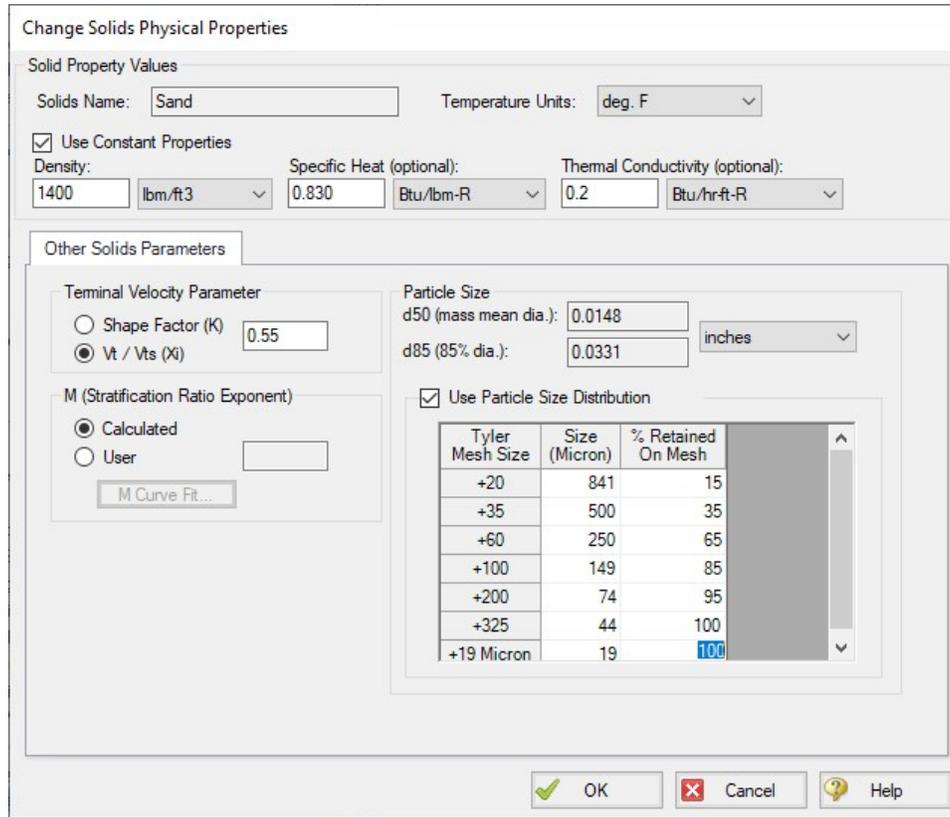


Figure 3: Solids Library showing the definition of particle size distribution

Related Topics

SSL Engineering Assumptions.....	571
SSL Nomenclature.....	570

SSL Pump De-Rating

(SSL Module Only) When centrifugal pumps are pumping slurries the solids in the slurry can cause decreased pump performance. This is known as de-rating, as the process involves reducing the pump head curves provided by manufacturers which are based on clear water.

The SSL module offers three methods to estimate pump de-rating. The first is based on a standard published by the Hydraulic Institute in 2016. In AFT Fathom it is called the “[ANSI/HI Standard 12.1-12.6-2016](#)” method. The second is based on the Warman method. The third allows the user to define a CH, CQ and CE rather than having them calculated.

When the SSL module is used, a new tab appears on the [Pump Properties window](#) called Slurry De-Rating (Figure 1). Here the user can choose the de-rating method. The de-rating factors are shown in the

Output window Pump Summary as CH (for head correction) and CE (for efficiency correction). Figure 2 shows the Pump Summary with correction factors displayed. If you do not see them in your Output you can add them to the Pump Summary in the [Output Control window](#) opened from the Tools menu.

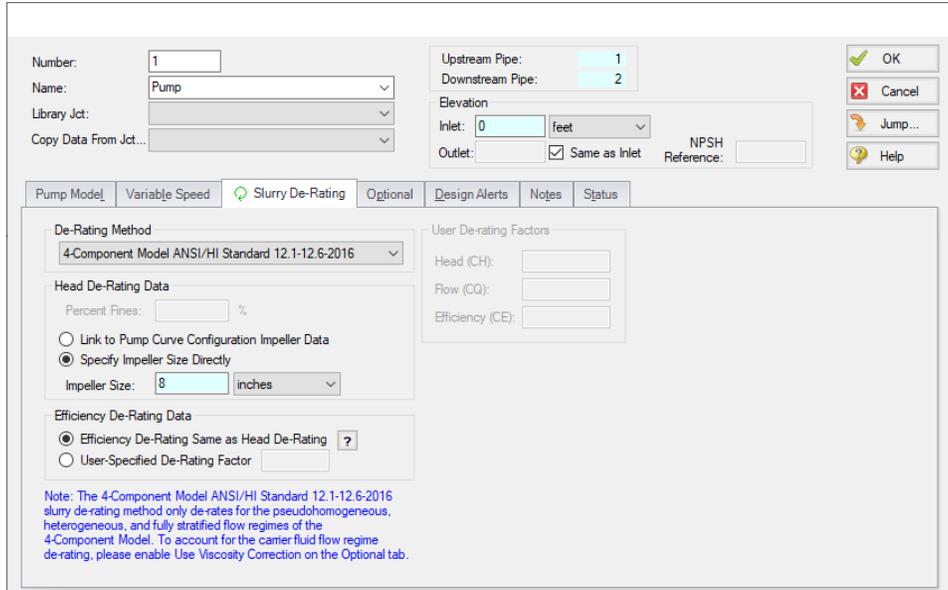


Figure 1: Slurry De-Rating tab on Pump Properties Window lets user estimate reduced pump performance due to solids

Jct	Results Diagram	Name	Vol. Flow (gal/min)	Mass Flow (lbm/sec)	dP (psid)	dH (feet)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (hp)	BEP (gal/min)	% of BEP (Percent)	NPSHA (feet)	NPSHR (feet)	CH	CE	CQ
2	Show ...	Pump	7.720	1.582	60.01	93.98	100.0	N/A	270.2	N/A	N/A	41.18	N/A	0.7917	0.7718	1.000

Figure 2: Pump de-rating factors CH and CE are shown in Output window Pump Summary

Theory of Pump De-Rating

The head reduction for slurry pumps is based on [ANSI/HI Standard 12.1-12.6-2016](#), page 52, Figure 12.3.2.7 (not shown here). Correction factors for solids density, solids concentration and percent of fine particles are applied per the standard. This figure and methodology is the same as found in Wilson, et al. 2006 on page 243.

The Warman Method is in use in industry but is not widely available. The method used in AFT Fathom is based on a white paper by [Walker, C. I.](#), Undated, page 2, and uses a correction factor for impeller size. C_m is the solids mass concentration of the slurry in terms of percent mass.

$$CH = 1 - \left[0.000385 (S_s - S_f) \left(1 + \frac{4}{S_s} \right) \right] C_m \ln \left(\frac{d_{50}}{0.0227} \right)$$

The 4-Component Model option is a modified form of ANSI/HI Standard 12.1-12.6-2016 by [GIW Industries Inc.](#) intended for use with the [4-Component Slurry Model](#). The head reduction is given by the following equation.

$$r_h = \sqrt{(r_{h,f})^2 + (r_{h,p} + r_{h,h} + r_{h,s})^2}$$

Where r_h is the total percent head reduction factor (i.e. 1 percent de-rate is 1.0) and $r_{h,f}$, $r_{h,p}$, $r_{h,h}$, and $r_{h,s}$ are the contributions of the carrier fluid viscosity, pseudo-homogeneous fraction, heterogeneous fraction, and fully stratified fraction, respectively.

For viscosity below 20 centipoise, GIW guidelines state that $r_{h,f}$ can be assumed to be zero. Above 20 centipoise, [ANSI/HI 9.6.7-2021](#) should be used to calculate this term.

Note: The [Viscosity Correction](#) term $r_{h,f}$ is not automatically applied when pump slurry de-rating is in use. To incorporate this term, Viscosity Correction must be enabled in the Optional tab. That option alone determines whether $r_{h,f}$ is calculated and the recommended 20 centipoise threshold is not used.

The equations for the other components take the following general form where D_2 is pump diameter in meters, D_{REF} is 1.0 m, S_s is the solid specific gravity, S_x is the mixture specific gravity for each component, C_v is the volumetric concentration of solids, and X_x is the fraction of solids associated with each component. $S1$ is equal to 0.5 times D_2/D_{REF} and $S2$ is 0.4.

$$r_{h,x} = N \cdot 8(D_{REF}/D_2)^{S1} (d_{50x})^{S2} \frac{S_s - S_x}{1.65} \frac{X_x C_v}{0.15}$$

This results in the following three equations:

$$r_{h,p} = A \cdot 8(D_{REF}/D_2)^{S1} (d_{50p})^{S2} \frac{S_s - S_f}{1.65} \frac{X_p C_v}{0.15}$$

$$r_{h,h} = B \cdot 8(D_{REF}/D_2)^{S1} (d_{50h})^{S2} \frac{S_s - S_{fp}}{1.65} \frac{X_h C_v}{0.15}$$

$$r_{h,s} = C \cdot 8(D_{REF}/D_2)^{S1} (d_{50s})^{S2} \frac{S_s - S_{fhp}}{1.65} \frac{X_s C_v}{0.15}$$

Where A , B , and C are empirical parameters and d_{50p} , d_{50h} , and d_{50s} are the average particle sizes in mm of the pseudo-homogenous, heterogeneous, and fully stratified fractions, respectively. CH is determined from r_h with the following expression.

$$CH = 1 - \frac{r_h}{100}$$

Related Topics

SSL Engineering Assumptions..... 571
 SSL Nomenclature.....570

Related Examples

Beginner - Pump Sizing for Sand Transfer System - SSL 1092

SSL in Output

[\(SSL Module Only\)](#)

Selecting Slurry Output Data

The specific slurry data to be shown in the Output window is specified in the [Output Control window](#). Select the Slurry button on the Display Parameters tab to see the available parameters (Figure 1).

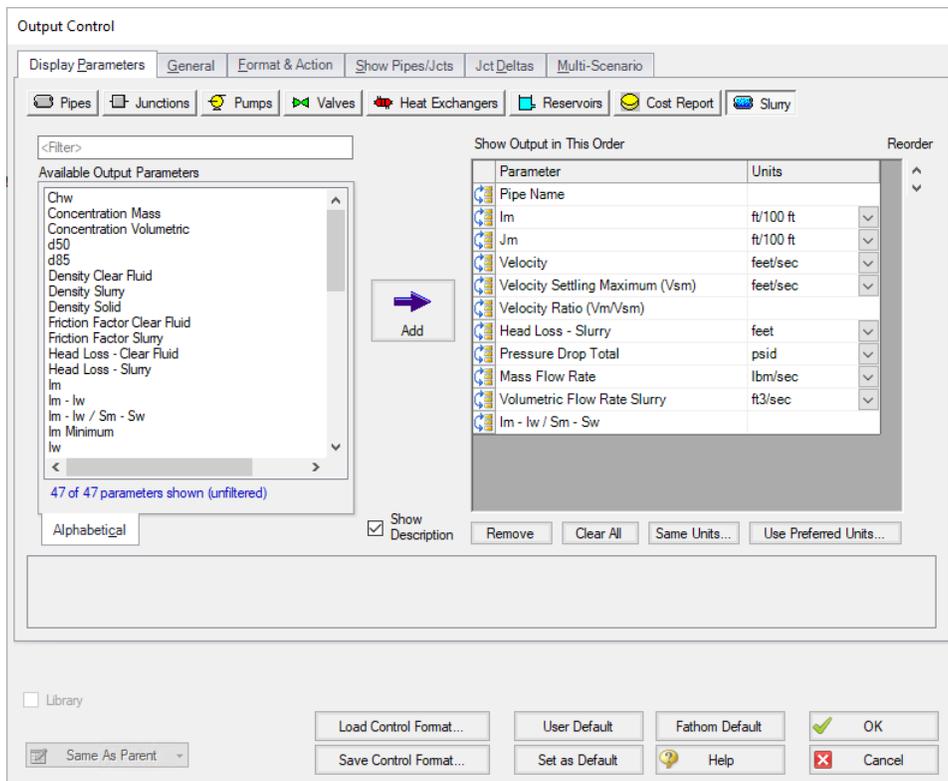


Figure 1: The Output Control window has a slurry button to allow choice of Output table parameters

Viewing Slurry Output

When an AFT Fathom model is run there is a special output table for the SSL module labeled Slurry that shows the results in the Pipes section (Figure 2).

Pipe	Name	Im (ft/100 ft)	Jm (ft/100 ft)	Velocity (feet/sec)	Settling Velocity Maximum (feet/sec)	Vm/Vsm	dH Slurry (feet)	dP (psid)	Mass Flow Rate (lbm/sec)	Vol Flow Slurry (gal/min)	(Im - Iw) / (Sm - Sw)
1	Pipe	14.17	9.608	21.90	11.48	1.908	1.922	1.228	1.582	7,720	0.08564
2	Pipe	14.17	9.608	21.90	11.48	1.908	48.040	30.683	1.582	7,720	0.08564
3	Pipe	14.17	9.608	21.90	11.48	1.908	4.804	3.067	1.582	7,720	0.08564

Figure 2: Output window showing Slurry table in Pipes section

Related Examples

Slurry System Feasibility Study - SSL (Metric Only).....1114

Related Topics

SSL Engineering Assumptions..... 571
 SSL Nomenclature..... 570

SSL in Graph Results

([SSL Module Only](#)) Slurry system curves are common way of interpreting slurry behavior. Such system curves can be specified in the Graph Control tab of the [Quick Access Panel](#).

When the SSL module is in use a new tab appears on the right called Slurry System Curve (Figure 1). Here, one can select the type of curve to generate.

Note: All pipes must be of the same diameter to use the Slurry System Curve.

Options to include pump curves (if a pump curve is used) and water curve cross plots are available.

The system curve is displayed in the Graph Results window (Figure 2).

Equations for Slurry System Curve

Fathom uses the equations below as are given in [Wilson et. al \(2006\)](#) on page 320 to calculate the velocity (V) and head gradient (j) for the slurry system curve.

$$V_x = \left[[0.22 V_{50}^M] 2gD / f_w \right]^{1/(M+2)}$$

$$j_x = [f_w/2gD]^{M/(M+2)} [0.22 V_{50}^M]^{2/(M+2)}$$

where V_{50} is the value of V_m at which 50% of solid are suspended in fluid, V_m is the mean velocity of the mixture, and f_w is the friction factor of water.

Since the head gradient depends on the velocity, all pipes must be the same diameter to produce a meaningful curve.

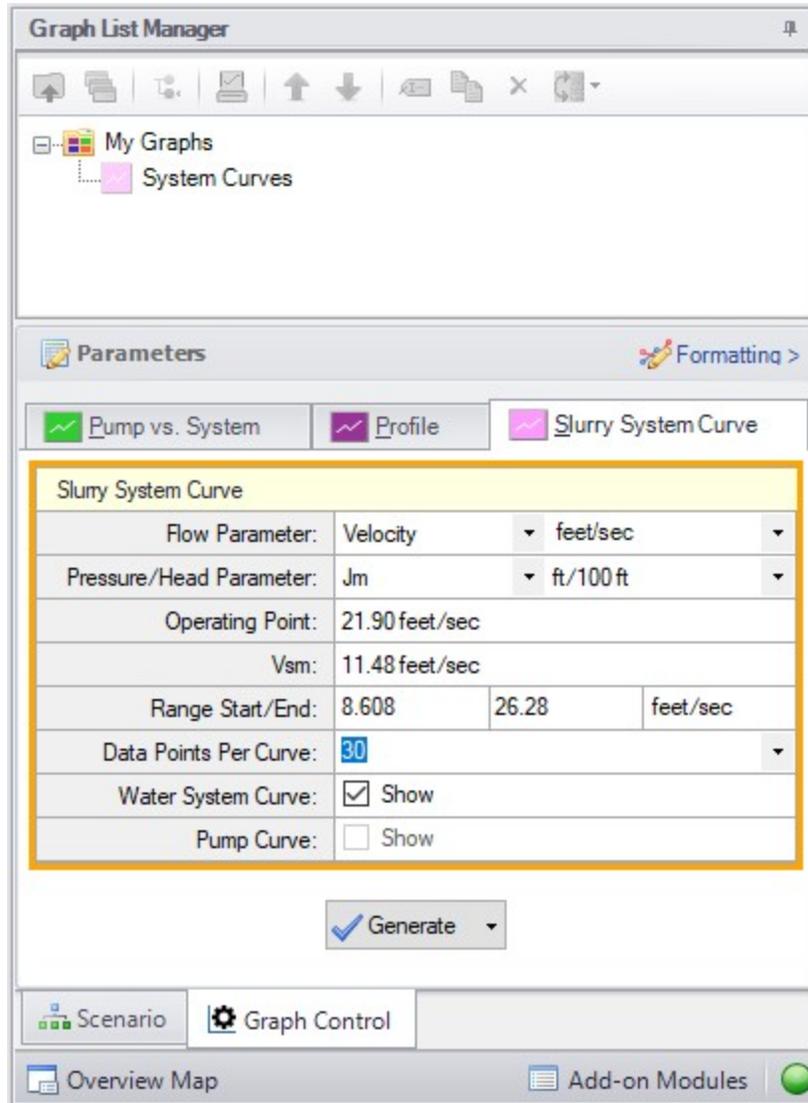


Figure 1: Slurry System Curve tab in the Quick Access Panel allows system curves to be created

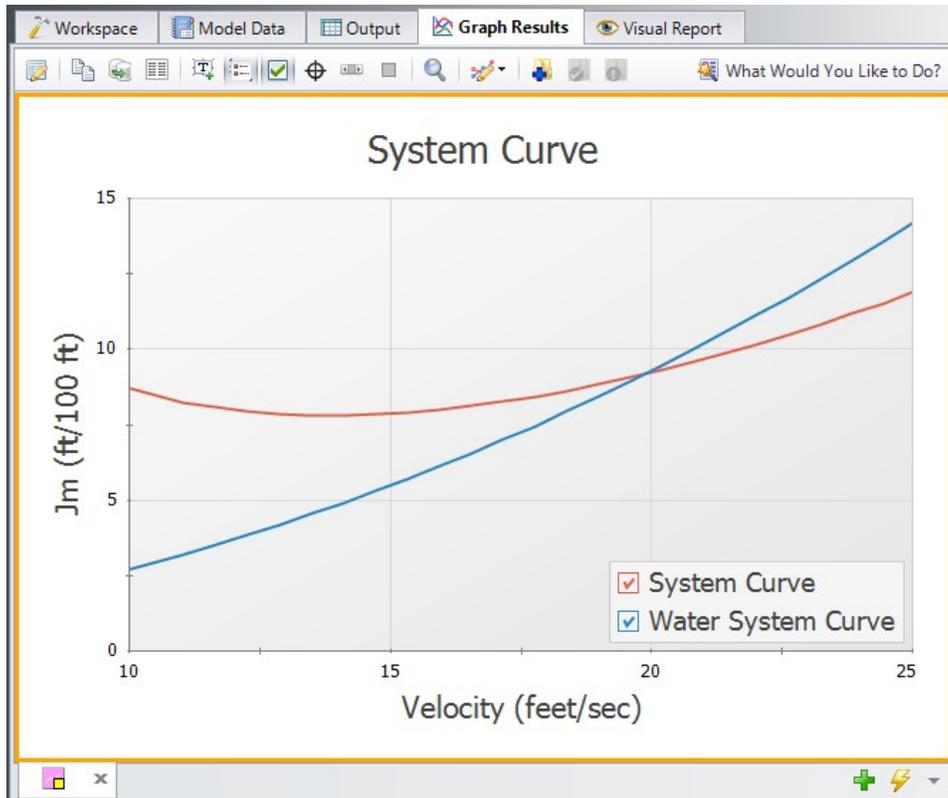


Figure 2: Graph Results window shows slurry system curve when using the SSL module

Related Topics

SSL Engineering Assumptions.....	571
SSL Nomenclature.....	570

Related Examples

Beginner - Pump Sizing for Sand Transfer System - SSL.....	1092
Slurries with Variable Fluid Properties - SSL.....	1102
Slurry System Feasibility Study - SSL (Metric Only).....	1114

SSL Initial Guesses for Pipe Velocity

(SSL Module Only) AFT Fathom uses [iterative methods](#) to solve for the equations of pipe flow. This requires initial guesses for pipe flowrate. Such guesses are optional for the user and can be entered on the Pipe Properties Window [Optional tab](#). If the user does not provide an initial guess then AFT Fathom makes an initial guess itself.

Typical settling slurry applications operate at a higher velocity than non-slurry flows and AFT Fathom will converge more reliably on a result if the initial guess at velocity reflects this higher velocity.

By default the initial guess flowrate for all pipes is based on a velocity of two times V_{SM} . This can be modified by the user in the [Advanced Parameters panel](#) or overridden directly by entering an initial guess for the pipes manually.

Related Topics

SSL Engineering Assumptions.....	571
SSL Nomenclature.....	570

SSL Troubleshooting

([SSL Module Only](#)) Figure 1 shows a system curve for a settling slurry. One of the distinct aspects of settling slurry behavior is the minimum point of the system curve. This means that for a given j_m (or, in general, head difference) there are two potential flowrates, as shown in Figure 1 below. This has the potential for unstable numerical results.

It is recommended that the system operate with a velocity ratio (V_m/V_{sm}) somewhat greater than 1.0 in all pipes so that solids will not settle out of the carrier fluid and form a stationary bed (which is not modeled using the SSL module) or plugging the pipe.

Often it is common practice to operate at or above the minimum i_m/j_m point to achieve a desirable velocity ratio, and to avoid higher head losses that can occur to the left of the i_m/j_m minimum. For example, in the below figure the minimum i_m occurs at about 5.25 m/s, so a mean velocity of 5.25 m/s or greater would be favorable. Model convergence may be difficult the closer the velocity ratio is to 1.0. Note that head loss curves like the one shown below will vary at different values of pipe diameter and solids concentration.

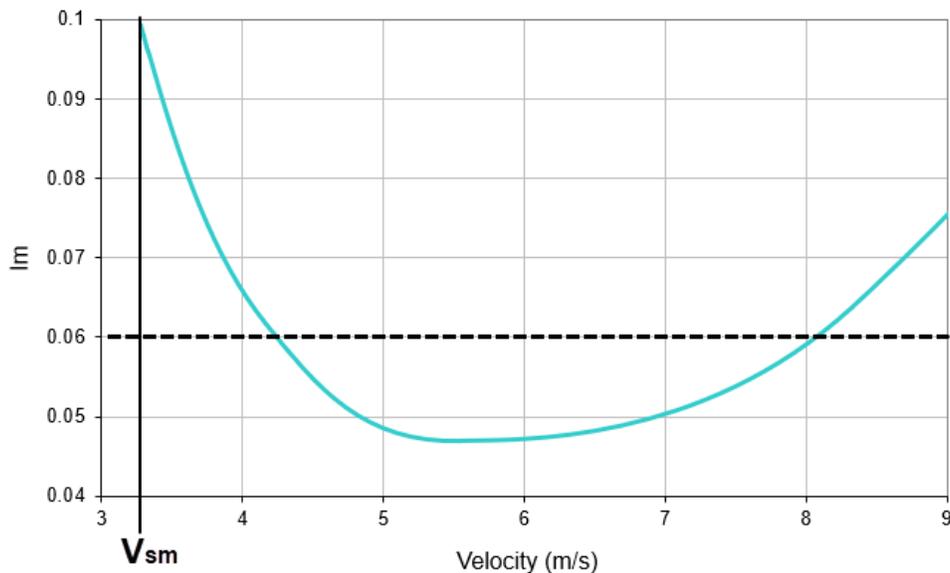


Figure 1: i_m vs mean velocity for a slurry system at a specific slurry concentration and pipe diameter.

Related Topics

SSL Engineering Assumptions.....	571
SSL Nomenclature.....	570

Tips When Using GSC with SSL Module

The [GSC module](#) in AFT Fathom can be used in conjunction with the [SSL Module](#). Similar considerations apply as are discussed in [SSL troubleshooting](#). If one is looking for a goals but the system is not a fixed flow type system then the goal seeker can become unstable.

The following tips are the same as given for general [GSC troubleshooting](#) but can be even more important when modeling slurries:

- Use bounds on design variables to keep the search within certain limits
- Vary the design variable starting point. If the simulation is started far away from the actual value then GSC may not be able to find the solution
- Manually modify some of the Numerical Control parameters in the [Goal Seek & Control Group](#). Specifically, increasing the Relative Finite Difference Step Size can help the goal seeker obtain better gradients and provide more reliability

Related Topics

SSL Engineering Assumptions.....	571
SSL Nomenclature.....	570

SSL Pipe Slope

[\(SSL Module Only\)](#) Pipes with different slopes will have different pressure drops for slurry flow when using the [Wilson Addie Clift](#) or [4-Component](#) settling slurry models because of the way solids are carried horizontally, vertically or in between. AFT Fathom determines the pipe slope based on the pipe length and connected junction elevations.

Pipe slope corrections are not applicable to the [Wasp-Durand Heterogeneous Slurry Model](#).

The pipe angle can be added to the Slurry output in the [Output Control table](#).

Related Topics

Sloped Pipe Theory For Slurries.....	568
--------------------------------------	-----

XTS - Extended Time Simulation

What is the XTS module?

An optional add-on module to AFT Fathom which allows users to simulate system transient behavior.

While steady-state modeling answers many design questions, some questions cannot be adequately answered without considering how systems behave over time. Questions such as how long it will take to fill a tank require a dynamic system model. The AFT Fathom XTS module answers such questions.

The XTS module allows you to model transient system behavior. Users can specify the time duration of the simulation, time step size, control system parameters, and how components such as pumps and valves operate over time. Operations such as valve position changes can occur during a specified time schedule, or can occur in response to events in the system thereby simulating control system actions.

Users can also specify tank volume on [“finite” tanks](#), so that tank draining and filling can be simulated. Tanks can be open or pressurized, with the gas pressure automatically calculated as the liquid level changes.

The XTS module is a powerful tool which extends AFT Fathom’s powerful modeling capabilities into the dimension of time.

Note: Transient modeling with XTS does not support heat transfer.

How Does the XTS Module Work?

The XTS module can be described as using a lumped or quasi-steady approximation to time simulation. This means it represents the transient system behavior as a sequence of pseudo-steady-state solutions. In between each pseudo-steady-state solution it adjusts transient parameters, performs mass balances on tanks, and changes component operations as specified by the user.

As a thought experiment, consider a system which you want to simulate for ten minutes in one minute increments. This would require eleven time step solutions (time zero and each minute up to ten). You could manually do this with standard AFT Fathom by running a steady-state model eleven times, and in between each run adjusting the input parameters for the next run based on the results of the previous run. The XTS module automates this manual process.

Besides the automation benefits, the XTS module offers additional benefits such as transient output data management, consolidation and display, and graphing tools to review the transient results.

Using the XTS Module

The user has the option of activating or not activating the XTS module when AFT Fathom first loads. After AFT Fathom is loaded, the XTS module can be activated or deactivated for use from the [Modules panel](#) in the Analysis Setup window or the Tools menu. Whether or not XTS is activated impacts the Analysis menu, Analysis Setup window, Status Bar and multiple other AFT Fathom functions as is discussed in the Modules panel topic.

If the XTS module is active, the user can still run models in Steady Only mode by selecting this under Time Simulation on the Analysis menu. Hence there are three possibilities for XTS:

- XTS is not active
- XTS is active and operated in Steady Only mode
- XTS is active and operated in Transient mode

Types of Transients Modeled

- **Reservoirs -**
 - Reservoirs can be modeled as [finite tanks](#) with user specified tank geometry.
 - Finite reservoirs with known geometry allows AFT Fathom to model liquid level changes over time to simulate tank filling and draining and its impact on the flow in the system.
 - Finite reservoirs can be open or enclosed. If enclosed, then surface gas pressure changes are automatically updated.
- **Pump transients -**
 - [Pumps](#) can be started or stopped based on [time or event transients](#)
 - Pump and [Pump as Turbine](#) speed can be changed with time
 - Pump flow or head can be changes with time
 - Controlled pump setpoint can be changed with time
 - Pump as Turbine [Load Rejection](#)
- **Valve transients -**
 - [Valves](#) can be closed or opened based on [time or event transients](#)
- **Control Valve transients -**
 - [Control valve](#) setpoints can be varied based on [time or event transients](#)
- **Other junction transients -**
 - [Spray Discharge](#) junctions can be closed or opened based on [time or event transients](#)
 - [Assigned Flow](#) junctions can have flow varied based on [time or event transients](#)
 - [Assigned Pressure](#) junctions can have pressure varied based on [time or event transients](#)
 - [Reservoir](#) junctions can have liquid level or surface pressure varied based on [time or event transients](#)
 - [Three-way valve](#) junctions can have valve position varied based on [time or event transients](#)

Related Topics

Modules Panel.....	85
Add-on Modules.....	538
Transient Control Group.....	143
Simulation Duration Panel.....	143
Choosing Time Simulation.....	588
Junction Transient Data.....	588
Finite and Infinite Reservoirs.....	601
XTS in Model Data.....	611
XTS in Output.....	612
XTS in Graph Results.....	613
XTS in Workspace.....	610

Related Examples

Beginner - Filling a Tank - XTS	1038
Reservoir Balance - XTS	1050
Valve Closing - XTS	1058
Housing Project - XTS	1072
Variable Demand - XTS	1081
Controlled Heat Exchanger Temperature - XTS	1066
Fixed Head Supply Tank - GSC XTS	1263

Choosing Time Simulation

[\(XTS Module Only\)](#) When using XTS, the option Time Simulation appears in the Analysis Menu. This drops down to two menu choices: Steady Only and Transient (Figure 1). Users can thus build steady models with XTS or transient models.

The Transient option requires the [Transient Control Group](#) to be defined in addition to the other typical settings. The Steady Only option makes the Transient Control Group settings optional.

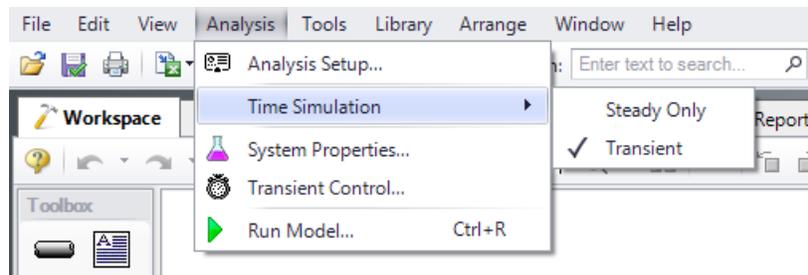


Figure 1: Time Simulation option in the Analysis menu

Related Topics

Junction Transient Data	588
Finite and Infinite Reservoirs	601
XTS in Model Data	611
XTS in Output	612
XTS in Graph Results	613
XTS in Workspace	610

Junction Transient Data

[\(XTS Module Only\)](#) Many junction types can have active transient data specified. The junction types are listed in Table 1:

Table 1: Junctions that support transient data

Junction Type	Type of Transient
Assigned Flow	Flow rate
Assigned Pressure	Pressure
Branch	Flowrate source or sink
Control Valve	Setpoint
Pump	Speed, flowrate, head rise, control setpoint
Reservoir	Liquid level, surface pressure
Spray Discharge	Flow area
Three-Way Valve	Position
Valve	K , C_v , or open percentage

The active transient data is entered on the Transient tab (Figure 1). The type of data that can be entered for each junction type is listed in Table 1. If a junction does not have any active transient behavior, there is no need to enter any data on the Transient tab.

In most cases, transient data can be entered as absolute values or as a percentage of steady-state. The distinction is made when choosing between Absolute Values and Relative to Steady-State Value. Some junctions may have additional options, such as the Valve junction (Figure 1).

The first data point always needs to match the steady-state value which is usually entered on the tab at the far left. If, for example in Figure 1, the steady-state data for C_v of 260 is changed to 200, the transient data will also need to be changed. However, if in Figure 1 the Relative option is specified, the first data point would be 100%, and when the C_v is steady-state is changed from 260 to 200 there is no need to change the transient data since the first data point is still 100%. But it is now relative to a different steady-state value.

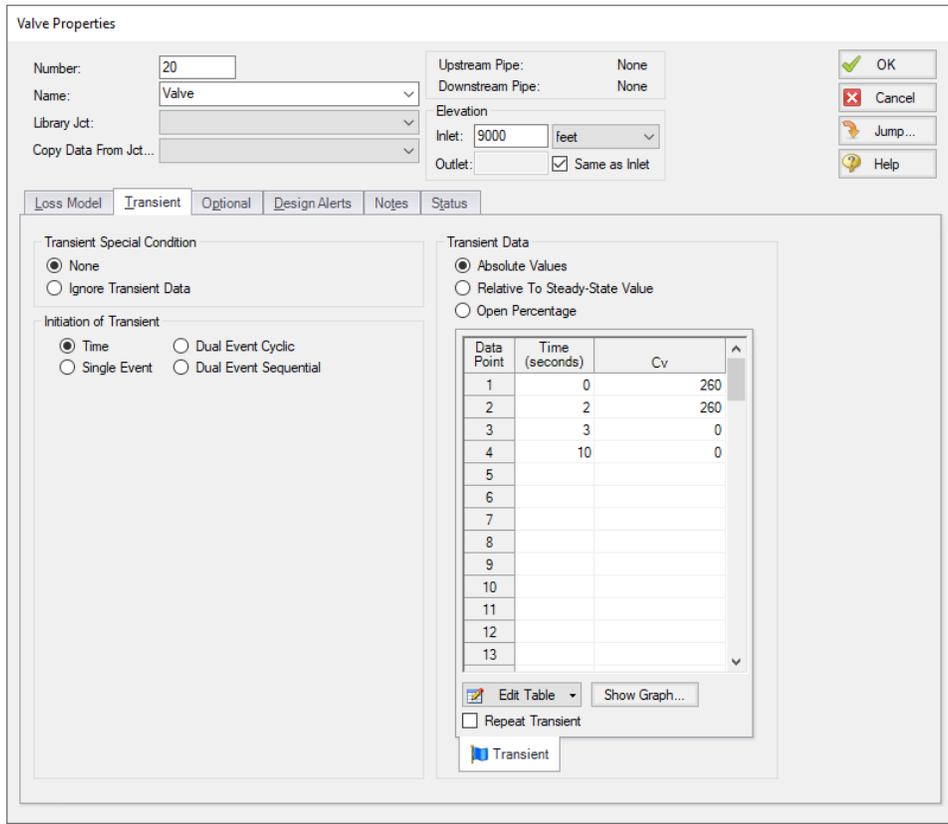


Figure 1: Example of junction transient data

Repeat Transient

If the transient data is periodic, you can enter the data for one cycle of the period and then tick the check box for Repeat Transient. This will cause the one cycle of transient data to be repeated once it has reached the end. The repetition will continue until the end of the simulation.

Transient Special Conditions

There may be occasions where, once having entered the transient data, you do not wish the transient to activate for a particular case. One option is to just delete the transient data. A second option is to specify the Transient Special Condition as Ignore Transient Data. In such a case, the data can be left in the window for future use, but will not activate during the run.

In Figure 1, there is an area called Initiation of Transient, and the selection is Time. This means that the specified transient is linked to the time of the simulation. Specifically, the valve transient data is activated as soon as the simulation is started. Because the transient data is constant for one second, the valve does not start to close until the simulation time reaches one second.

Transients can also be initiated based on events. For instance, we can change the valve in Figure 1 to start closing when the pressure at some other system location reaches a certain level. Or maybe when a tank has filled up and thus reached a certain level. In such a case, the time zero data in Figure 1 does not refer to the start of the simulation, but is relative to when the event occurs. If, for instance, the pressure never reaches the specified level, the valve never closes.

Related Topics

Initiation of Transients.....	591
Multi-Condition Events.....	596
XTS Special Conditions.....	600
Choosing Time Simulation.....	588
Finite and Infinite Reservoirs.....	601
XTS in Model Data.....	611
XTS in Output.....	612
XTS in Graph Results.....	613
XTS in Workspace.....	610

Related Examples

Valve Closing - XTS.....	1058
Housing Project - XTS.....	1072
Variable Demand - XTS.....	1081
Controlled Heat Exchanger Temperature - XTS.....	1066

Initiation of Transients

[\(XTS Module Only\)](#) In general, transients can be initiated according to a certain time or according to certain events that happen in the system. These two categories are referred to as time based and event based transients. This is defined in the *Initiation of Transient* area of a junction's transient tab.

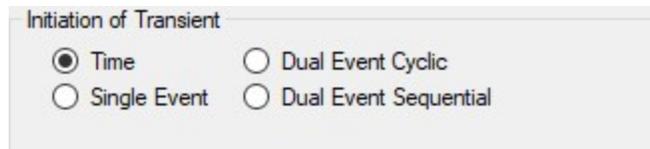


Figure 1: Type of transient initiation

It is helpful to consider the different time scales that are used when referring to the initiation of a transient, as shown in Figure 2.

Absolute and Simulation Time are often identical. The simulation Start Time is set in [Simulation Duration panel](#) to an Absolute Time of zero seconds by default, but it could be non-zero. In Figure 2, the Simulation Start Time is set to a value of 2 seconds – the entire Simulation Time timeline is shifted forward 2 seconds.

Event Time refers to the time data defined in Transient Data. This could be equal to Simulation Time (and Absolute Time) depending on how the event is defined. For example, all three time bases will match for a pump trip event starting at the beginning of a simulation with Start Time equal to zero. However, the event may not start until a certain system condition is met and zero Event Time could be well into the simulation.

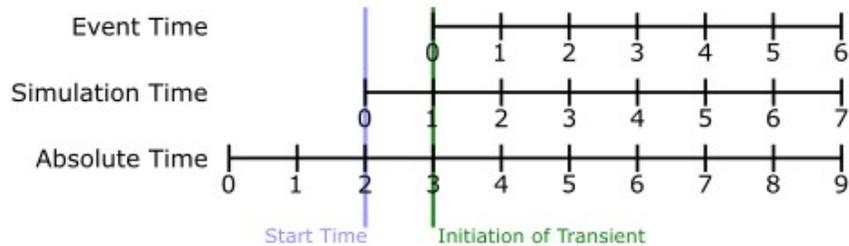


Figure 2: Time basis

Time

Time based transients modify a parameter directly according to Simulation Time.

For example, consider a valve closure transient. The valve starts to close at two seconds into the simulation, and the time it takes for the valve to close is one second. After that, the valve stays closed. To define this event with the Time option, the Start Time must be set to zero, and the Transient Data must look something like the data shown in Figure 3.

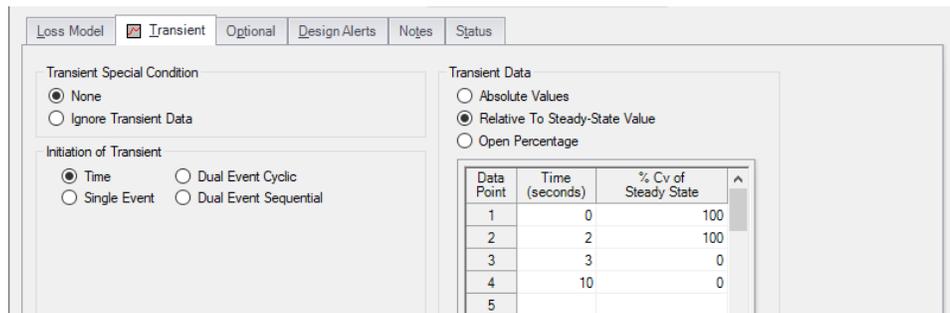


Figure 3: Defining a valve closure at a specific time

Single Event

Single Event initiated transients do not begin until a certain condition is met.

To completely define an Event, four parameters are required:

- **Event Type** - What parameter is being considered to initiate the transient. See Table 1 below for a list of the possible Event Types.
- **Condition** - The logical relationship between Event Type and Value (e.g. greater than, less than, etc)
- **Value** - The specific magnitude of the Event Type parameter at Location
- **Location** - Where in the model the Value should be observed

Given the same valve closure profile described in Figure 3, we can set the valve to remain open until (and if) the pressure at the inlet of P101 exceeds 100 psia. Note that the Transient Data is identical - but the initiation of the event may happen at a very different time.

Data Point	Time (seconds)	% Cv of Steady State
1	0	100
2	2	100
3	3	0
4	10	0
5		
6		
7		

Figure 4: Event definition

Table 1: Event types

** Multi-Condition **	Kv	Reservoir Liquid Height
Check Valve State	Mass Flow Rate in Pipe	Reservoir Liquid Volume
Control Valve State	Pressure Stag. Diff. at Junction	Reservoir Surface Elevation
Cv	Pressure Stag. Diff. Between Pipes	Reservoir Surface Pressure
EGL at Pipe	Pressure Stagnation at Pipe	Spray Discharge Outflow
EGL Diff. at Junction	Pressure Static at Pipe	Time Absolute
EGL Diff. Between Pipes	Pressure Static Diff. at Junction	Time Difference
HGL at Pipe	Pressure Static Diff. Between Pipes	Valve Open Percent
HGL Diff. at Junction	Pump Speed	Velocity in Pipe
HGL Diff. Between Pipes	Relief Valve State	Volumetric Flow Rate in Pipe

Dual Event Cyclic

The Dual Event options, as the names imply, require the definition of two events. The definition for each event follows the same structure as a Single Event.

Dual Event Cyclic will continue monitoring both triggering events throughout the simulation and will apply the corresponding transient data when the conditions are met.

To continue with the valve example, we can define our valve not only to open when a pressure is exceeded, we can tell it to close when the pressure drops below a certain setting.

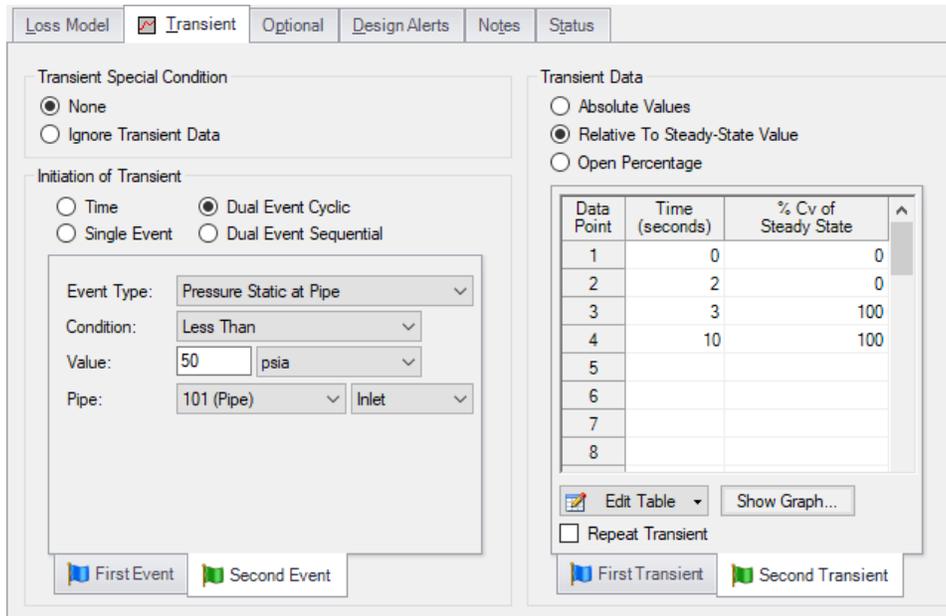


Figure 5: Dual Event set up. Note that a Second Event requires additional data for the Second Transient

Dual Event Sequential

Dual Event Sequential is nearly identical to Dual Event Cyclic. In fact, all of the required user input is the same.

The difference is that the two defined events are only applied once. That is – after the second event is complete, no more transients will be applied even if the conditions are met.

Junctions With Inherent Event Logic

There are two junction types in AFT Fathom which have built-in, or inherent, event logic. These are the check valve and relief valve. The user does not need to specify the nature of the events, and in fact is not allowed to.

The inherent event logic is very similar to the Dual Event Cyclic logic described previously.

- **Check valve** - The [check valve](#) has two built-in events. The first is that it closes when backflow starts to occur. The second is that it reopens when sufficient pressure differential occurs. These transients are assumed to be instantaneous.
- **Relief valve** - The [relief valve](#) has two built-in events. The first is that it opens when the set pressure is reached. The second is that it closes again when the pressure falls back below the set pressure. These transients are assumed to be instantaneous unless the Pressure/Pressure Opening/Closing profile is used.

Related Topics

Junction Transient Data	588
Multi-Condition Events	596
XTS Special Conditions	600

Related Examples

Valve Closing - XTS	1058
Housing Project - XTS	1072
Variable Demand - XTS	1081
Controlled Heat Exchanger Temperature - XTS	1066

Multi-Condition Events

One of the [Event Types](#) available in XTS is Multi-Condition. This feature allows starting a transient based on more than one parameter in the model.

For example, it may be desirable to open a valve when either of two pressures is exceeded.

To define a Multi-Condition Event, first select Multi-Condition as the Event Type from the junction's Transient tab.

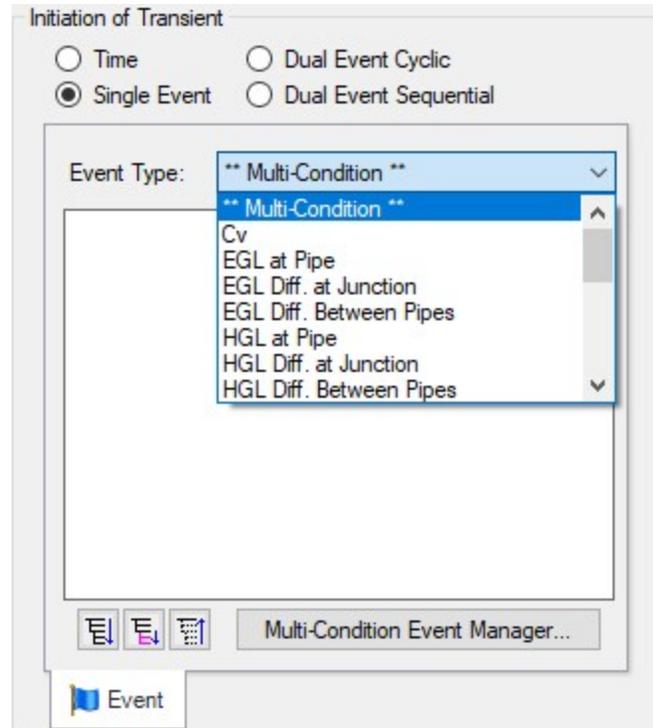


Figure 1: Selecting Multi-Condition Event Type

Event Manager

Clicking Multi-Condition Event opens the Event Manager, where Simple Events and Multi-Condition Events can be defined.

The Event Manager defines events model-wide. That is, if the active scenario is changed, the same events will be accessible.

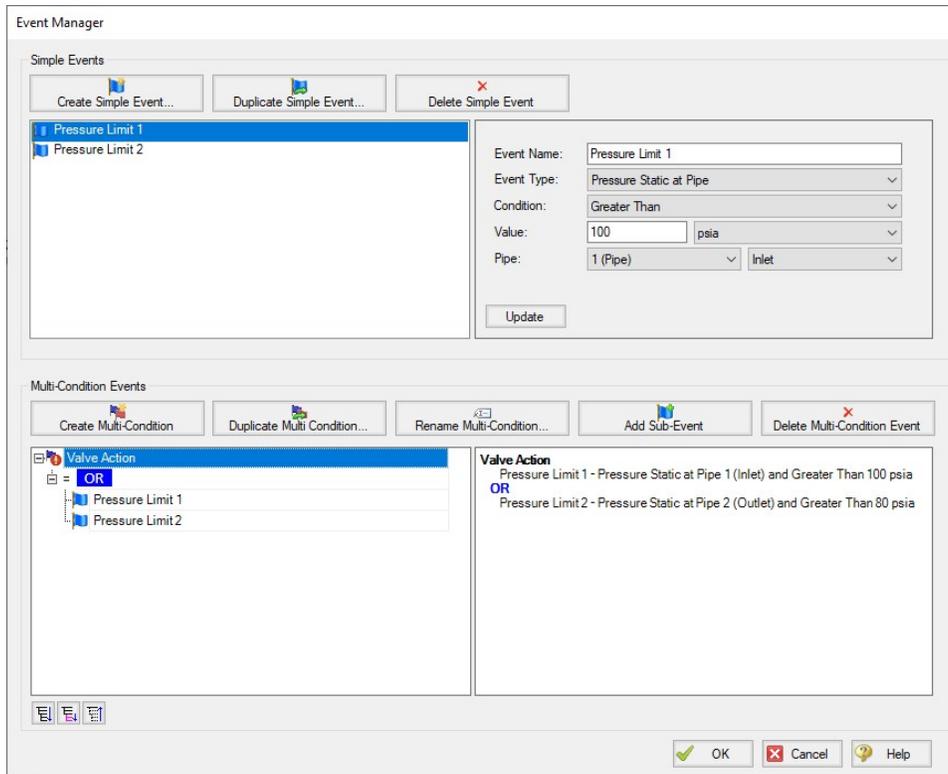


Figure 2: The Event Manager

Simple Events

The Event Manager allows for the definition of Simple Events and Multi-Condition Events.

Simple Events are nearly identical to normal (non Multi-Condition) events defined in a Transient tab. They require a Parameter, Condition, Value, and Object as well as a unique Name.

Clicking "Create Simple Event..." adds a new Simple Event to the list with default values. These values should be updated with the desired values and applied with the Update button. Simple Events can be duplicated to help reduce repeated data entry.

Multi-Condition Events

Multi-Condition Events are a collection of other Events with a logical condition applied to them. When creating a new Multi-Condition Event, a prompt for a Name appears. After entering a name, an empty Multi-Condition is created with a default logical condition of AND. This logical condition can be:

- **AND** - All of the Simple Event conditions must be met for the Multi-Condition Event to trigger.
- **NOT** - The Multi-Condition Event triggers when none of the conditions evaluate to true.
- **OR** - One or more of the Simple Event conditions must be met for the Multi-Condition Event to trigger.

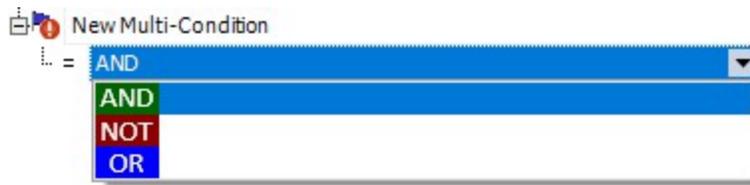


Figure 3: Conditional logic for Multi-Condition Events

Any number of Simple Events can be added to a Multi-Condition Event as Sub-Events:

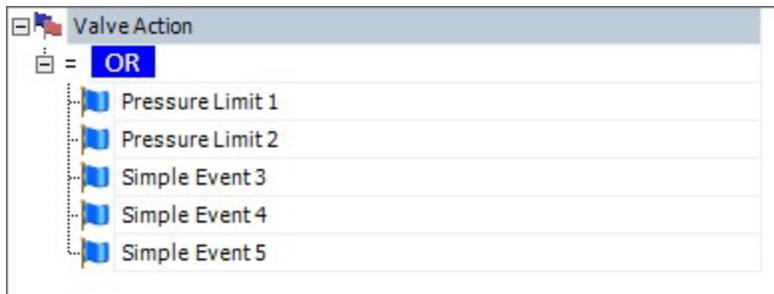


Figure 4: Multiple Events

Complex Multi-Condition Events

More complex events can be defined by adding one or several Multi-Condition Events as Sub-Events to another Multi-Condition Event.

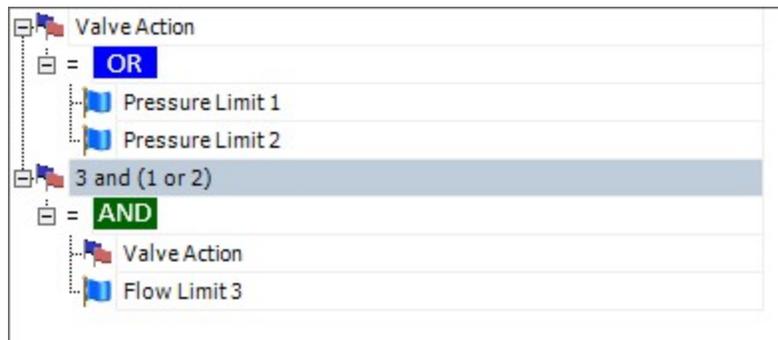


Figure 5: A Multi-Condition Event containing another Multi-Condition Event (trigger if Limit 3 AND (Limit 1 OR Limit 2))

Using Multi-Condition Events

All Multi-Condition Events are listed in the junction's Initiation of Transient section. Only one Multi-Condition Event can be selected as the Initiating Event for any particular Transient Data.

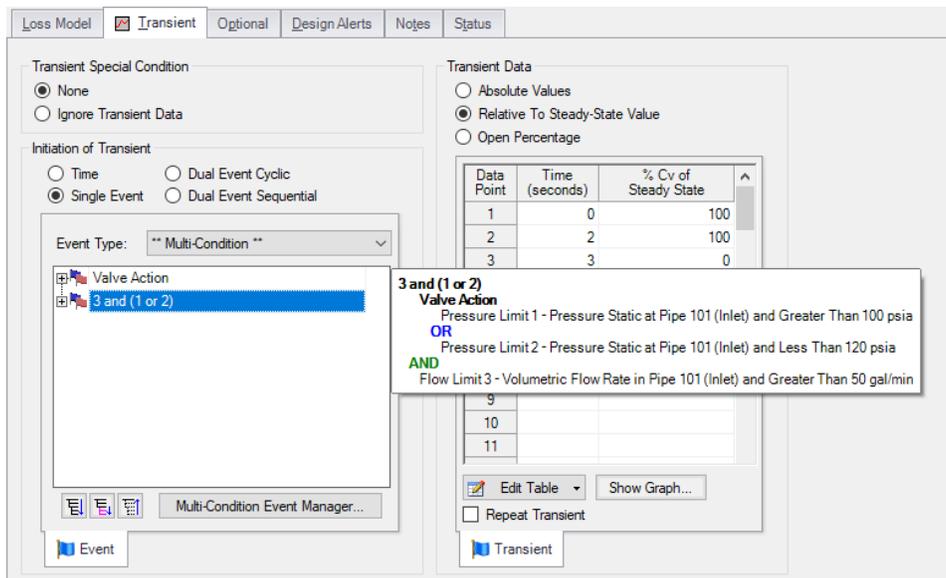


Figure 6: The selected Multi-Condition Event controls the trigger of the Transient Data. Hovering over the Multi-Condition events here shows an expanded definition of the event.

Related Topics

Junction Transient Data	588
Initiation of Transients	591
XTS Special Conditions	600

XTS Special Conditions

([XTS Module Only](#)) Some junctions have [Special Conditions](#) that alter the normal state of the junction. The junctions that have special conditions set are shown using a special symbol before the ID number (an “X” by default). This symbol can be customized in the [User Options](#) window.

Special Conditions with No Transient Data

If a Special Condition is applied to a junction without any transient data, then the transient solution will retain the Special Condition. For instance, if a Special Condition is set for a valve junction, the steady-state solution will solve with the valve closed. In addition, the transient solution will run the entire simulation with the valve closed as well because there is no transient data entered which implies the valve state is not intended to change.

Special Conditions with Transient Data

If a Special Condition is applied to a [junction that has transient data](#), then the transient solution will ignore the Special Condition. It is then incumbent upon the engineer to relate the Special Condition to the initial transient data.

For instance, assume the user wants to simulate the transient that occurs during a valve opening. The steady-state solution will have the valve closed, and this is modeled by using a Special Condition for the valve junction. In such a case, the C_v data entered on the valve junction's Loss Model tab will be ignored.

On the Transient tab, a transient can be entered. The first data in the table should then be $C_v = 0$, which corresponds to a closed valve. From there the C_v can be increased above zero as desired to open the valve.

If the user were to keep the C_v equal to zero, then that would be equivalent to the previous case where the user set the Special Condition but did not enter any transient data.

Pump Special Conditions

Pump Special Conditions are slightly more complicated than for valves. In fact, [pump junctions](#) have two types of Special Conditions. The first is to turn the pump off and have no flow through it. This is called “**Pump Off No Flow**”. The second type of Special Condition turns the pump off but allows flow to go through the pump. This is called “**Pump Off With Flow Through**”.

Why would one want to use one Special Condition rather than the other? The simple answer is that the first Special Condition is more appropriate for positive displacement type pumps and the second for centrifugal pumps.

Here's why. When a positive displacement pump is turned off and has a pressure difference across it, it will usually act like a closed valve and not allow flow to go through it. Thus the first Special Condition would be most appropriate. For instance, assume one wants to model the transient that occurs during the startup of a positive displacement pump. The best way to do this would be to use the first Special Condition with no flow, and then input a flowrate transient in which the first data point is zero flow.

On the other hand, when a centrifugal pump is turned off and has a pressure difference across it, in the absence of other valving which prevents flow the pump will usually allow flow to go through it. Thus the Special Condition that allows through flow would be most appropriate. For example, assume one wants to model the transient during the startup of a centrifugal pump. One would use the second Special Condition, and, in conjunction with a pump curve entered at 100% speed, input a speed transient with the initial speed as zero.

Transient Special Conditions

Junctions which accept transient data also support Transient Special Conditions. The default for all junctions is None. The other choice is Ignore Transient Data. This allows one to have no transient initiation at that junction without having to delete the transient data.

When a Transient Special Condition is set to Ignore, a # symbol is displayed adjacent to the junction number on the Workspace. This symbol can be customized in the [User Options](#) window.

Related Topics

Junction Transient Data	588
Initiation of Transients.....	591
Multi-Condition Events.....	596

Finite and Infinite Reservoirs

Without the [XTS Module](#), reservoirs are infinite. With the XTS module, reservoirs can be infinite or finite.

- **Infinite reservoirs** - infinite reservoirs refer to a massive body of fluid whose surface level does not change appreciably as a result of liquid inflow or outflow during the time frame of the simulation. An example is a large lake or the ocean.
- **Finite reservoirs** - ([XTS Module Only](#)) finite reservoirs (also called finite tanks) refer to a body of fluid which is small enough that its surface level changes significantly during the time frame of the simulation as a result of liquid inflow or outflow. An example is a tank which drains as the simulation progresses.
 - Finite reservoirs can be **open** or **closed**. Finite open reservoirs are open to the atmosphere or some other fixed surface pressure. Typically open reservoirs have a constant surface pressure with time, although users have the latitude to pre-specify time varying surface pressure.
 - Finite closed reservoirs are closed tanks in which, as the liquid level changes, the gas above the liquid expands or contracts and hence its pressure changes according to the gas thermodynamic laws. For closed reservoirs the user specifies the initial liquid level and gas pressure, but during the simulation all gas pressure and liquid levels are calculated by AFT Fathom.

Related Topics

Reservoir.....	333
Pipe X elevation of Y is below Reservoir Z tank bottom elevation.....	788
Tank Junction X began overflowing at Y seconds.....	782
Tank Junction X completely drained at Y seconds.....	782
Choosing Time Simulation.....	588
Junction Transient Data.....	588
XTS in Model Data.....	611
XTS in Output.....	612
XTS in Graph Results.....	613
XTS in Workspace.....	610

Related Examples

Beginner - Filling a Tank - XTS.....	1038
Reservoir Balance - XTS.....	1050
Housing Project - XTS.....	1072

Entering Reservoir Volume Data

([XTS Module Only](#)) When running AFT Fathom in Steady Only mode (or without the XTS module), each reservoir junction is considered an infinite reservoir. This means that the liquid heights and surface pressures entered are constant. When running transients, reservoirs can be either *infinite* or *finite*. Similar to Steady Only mode, an infinite reservoir maintains its level during the transient. The differences are discussed below and summarized in Table 1.

Table 1: Various possible inputs with infinite and finite Reservoir junctions

Tank Model	Tank Volume Needed	Surface Level/Pressure Required?	Surface Level Over Time	Surface Pressure Over Time
Infinite	No	Both	Constant or User Specified	Constant or User Specified
Finite	Open Yes	One or Both	Calculated	Constant or User Specified
	Closed Yes	One or Both	Calculated	Calculated from PV^n

Infinite Reservoirs

Infinite reservoirs refer to a massive body of fluid whose surface level does not change appreciably as a result of liquid inflow or outflow during the time frame of the simulation. An example is a large lake or the ocean. Figure 1 shows a Reservoir junction modeled as infinite.

Finite Reservoirs

Finite reservoirs (also called finite tanks) refer to a body of fluid which is small enough that its surface level changes significantly during the time frame of the simulation as a result of liquid inflow or outflow.

An example is a tank which drains as the simulation progresses. Tanks can have constant cross-section (e.g., a pure vertical cylinder) or tank cross-section can change with height. When the cross-section changes, the tank volume at each height data point must be specified.

Finite tanks can further be broken down into open or closed tanks.

- **Finite open tanks** - Finite open tanks are open to the atmosphere or some other fixed surface pressure. Typically open tanks have a constant surface pressure with time, although users have the latitude to pre-specify time varying surface pressure. Figure 2 shows a Reservoir junction modeled as a finite open tank.
- **Finite closed tanks** - Finite closed tanks are tanks in which, as the liquid level changes, the gas above the liquid expands or contracts and hence its pressure changes according to the gas thermodynamic laws. For closed tanks the user specifies the initial liquid level and gas pressure, but during the simulation all gas pressure and liquid levels are calculated by AFT Fathom. Figure 3 shows a Reservoir junction modeled as a finite closed tank.

Known Parameters Initially

In some cases the liquid level may not be known initially when modeling finite tanks. An example would be a vertical standpipe. At time zero the net flow at the standpipe would be balanced, and the liquid surface would find its own level (this would equal the local hydraulic gradeline). Hence the initial liquid level would be calculated based on equilibrium.

However, under transient conditions the liquid level will change, and the liquid volume in the standpipe can significantly affect pipeline behavior.

This is the purpose of the features in the Known Parameters Initially area (see Figure 2). By default, both the initial liquid level and initial surface pressure are assumed known. If you want to model one of them as unknown initially, select the desired parameter from the drop-down. If the surface pressure is selected as unknown initially, it functions similarly to the unknown liquid level except initial equilibrium surface pressure would be calculated.

This feature only affects the calculation at the initial “zero” time step. For all other time steps, the junction behaves the same as a junction in which the liquid level and surface pressure are known initially.

Cost Calculation

Reservoir Properties

Number: 1
Name: Lower Reservoir
Library Jct:
Copy Data From Jct...

OK
Cancel
Jump...
Help

Reservoir Model | Pipe Depth & Loss Coefficients | **Transient** | Optional | Design Alerts | Notes | Status

Tank Model
 Infinite Reservoir
 Finite Open Tank
 Finite Closed Tank

Known Parameters Initially
Both Parameters

Liquid Surface Pressure
0 psig

Liquid Surface Elevation:
10 feet

Pipe Elevation Pipe Depth
10 feet

Same Depth for All Pipes

Reference

Figure 1: Reservoir junction modeled as infinite

Cost Calculation

Reservoir Properties

Number:

Name:

Library Jct:

Copy Data From Jct...:

OK Cancel Jump... Help

Reservoir Model | Pipe Depth & Loss Coefficients | Transient | Optional | Design Alerts | Notes | Status

Tank Model
 Infinite Reservoir
 Finite Open Tank
 Finite Closed Tank

Known Parameters Initially

Cross-Sectional Area
 Constant Variable

Liquid Surface Pressure

Liquid Surface Elevation:
 Elevation Height from Bottom

Tank Height:

Pipe Elevation Pipe Depth

Same Depth for All Pipes

Tank Bottom Elevation:

Reference

Figure 2: Reservoir junction modeled as finite open

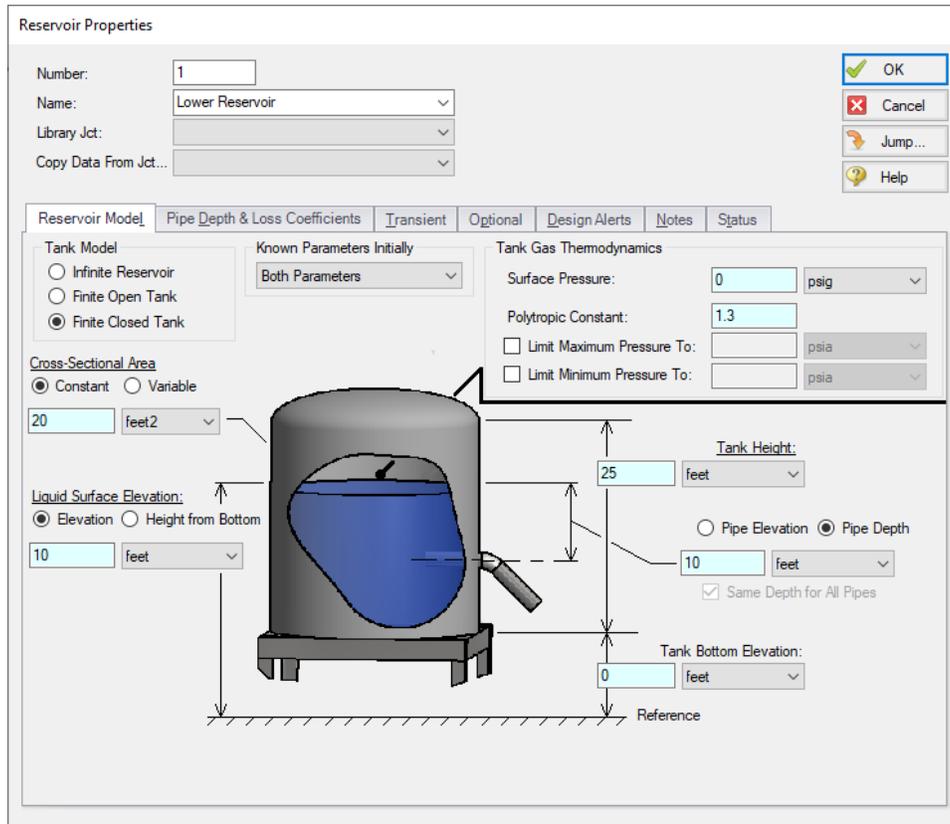


Figure 3: Reservoir junction modeled as finite closed

Related Topics

Entering Reservoir Transient Data	606
Finite Reservoir Maximum and Minimum Pressures	609
Finite Tank Overflow & Draining	608

Entering Reservoir Transient Data

[\(XTS Module Only\)](#)

Infinite Reservoirs

For an infinite reservoir, the liquid level or surface pressure can be varied over time according to a user specified transient profile (see Table 1 in the [Entering Reservoir Volume Data](#) topic). This is input on the Transient tab (see Figure 1).

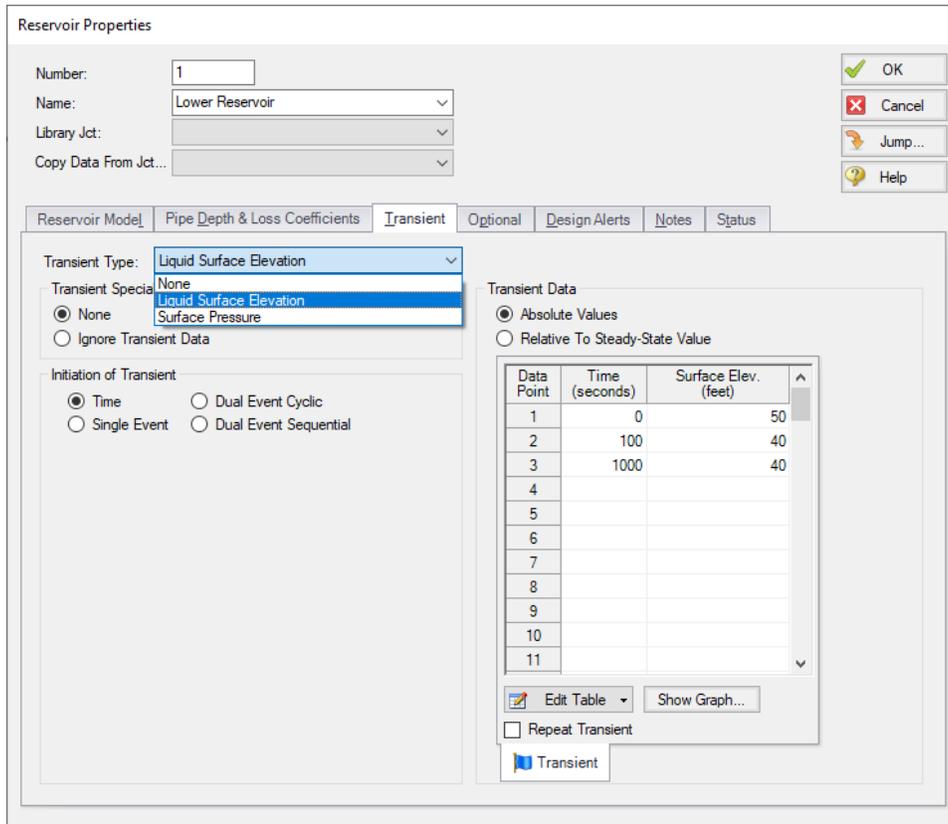


Figure 1: Infinite reservoirs can have user specified time varying liquid level or surface pressure

Finite Open Tanks

For a finite open tank, the surface pressure can be varied over time according to a user specified transient profile. This is input on the Transient tab (Figure 2).

Finite Closed Tanks

For a finite closed tank with a maximum allowable pressure (this is a relief pressure), the relief pressure can be varied over time according to a user specified transient profile. This is input on the Transient tab.

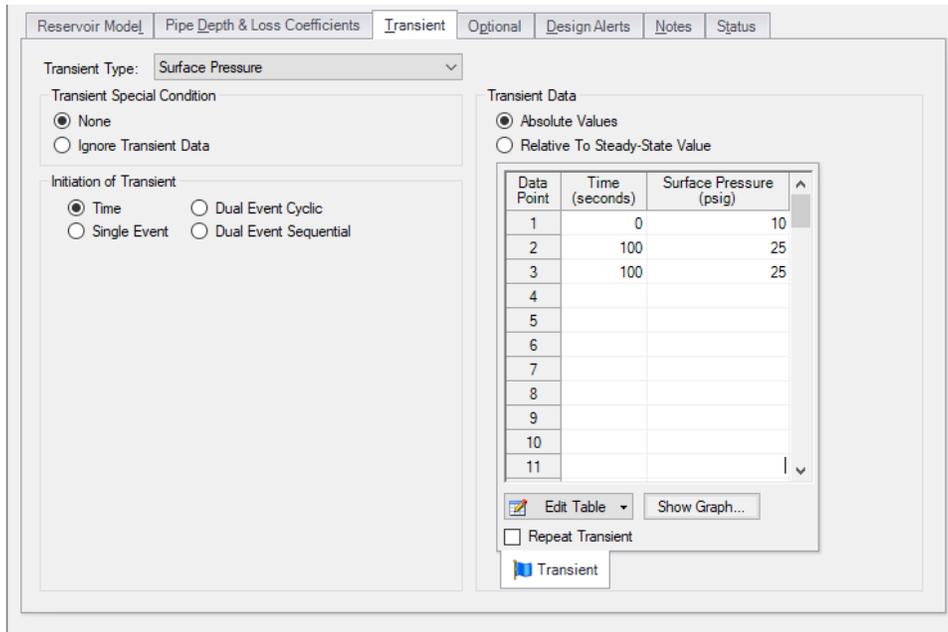


Figure 2: Finite tanks can have user specified time varying surface pressure

Related Topics

Entering Reservoir Volume Data	602
Finite Reservoir Maximum and Minimum Pressures	609
Finite Tank Overflow & Draining	608

Finite Tank Overflow & Draining

[\(XTS Module Only\)](#)

Finite Tank Overflow

As the transient progresses finite tanks can fill up. If the tank is open, then it is possible for the liquid to reach the top. How does AFT Fathom handle this?

When a finite tank liquid level reaches the top, it is assumed that the liquid spills over the top. Hence the liquid height is maintained at the top of the tank. Note that this causes a loss of mass from the system model.

If the tank is closed, the gas pressure will increase as the tank fills. Thus the tank cannot overflow. If the tank has a maximum pressure (which represents a relief line) then it is possible for the liquid mass to be lost as it would ostensibly flow out the relief line. The liquid level in the tank would then be maintained at the top of the tank.

Finite Tank Drained

When a tank drains to the bottom, it cannot supply any more liquid to the connected pipes. The system behavior after a tank has drained cannot be accurately modeled by AFT Fathom. In a real system, and in the absence of a valve to stop the flow, a drained tank which continues to flow would result in the connected pipes themselves draining and a gas/liquid interface moving down the pipes. AFT Fathom assumes all pipes are liquid full, and cannot model draining pipes.

Since it is not desirable in most applications to have the pipes drain, this limitation is not a significant issue. The user can, for example, use a valve in the pipe which closes when the tank drains. This would use an event transient.

Since AFT Fathom cannot model a tank after it has drained, what does it do? Rather than halt the transient run, which would not be very helpful to the user, AFT Fathom assumes the connecting pipe “turns off” when it no longer has liquid to supply it. To implement this, AFT Fathom sets the pipe’s Special Condition so that it turns the pipe off.

Finite Tank Pipes Uncovered

AFT Fathom allows multiple pipes to connect to a reservoir, all at different elevations. As a tank drains, pipes may be uncovered. AFT Fathom handles this in the same manner as described above for completely drained tank.

If a pipe is uncovered, then there are two possibilities:

- **Liquid flows from the pipe and into the tank** – AFT Fathom can model this case. The liquid free falls to the liquid surface, and appropriate boundary conditions are used to solve the system.
- **Liquid tries to flow from the tank and into the pipe** – AFT Fathom cannot model this case. Since the pipe is not covered by liquid, then the tank cannot supply liquid to the pipe and in reality the pipe itself would drain. AFT Fathom does the same thing as described above for a completely drained tank – it sets the pipe flow to zero.

Important: When a tank drains completely or sufficiently to uncover a pipe, AFT Fathom cannot adequately model the system behavior after this point unless a valve is used to stop the flow into the pipe. AFT Fathom will warn the user when this happens.

Related Topics

Entering Reservoir Volume Data	602
Entering Reservoir Transient Data	606
Finite Reservoir Maximum and Minimum Pressures	609

Finite Reservoir Maximum and Minimum Pressures

(XTS Module Only) A [finite closed tank](#) allows the user to specify maximum and minimum pressure. This is the pressure of the gas in the tank. The purpose of maximum and minimum pressure is to simulate relief and vent systems in the tank which represent over and underpressurization.

Related Topics

Entering Reservoir Volume Data	602
Entering Reservoir Transient Data	606
Finite Tank Overflow & Draining	608

XTS in Workspace

(XTS Module Only) The transient parameter labels on the [Workspace Layers](#) will change dynamically to be displayed at a selected time. The slider bar and buttons at the bottom of the Workspace function the same as the [Output window time selection controls](#). When the user changes the current time, displayed transient parameters are changed as well. In addition, color maps are updated when in use.

As shown in Figure 1, the current time is displayed in the bottom-left by the time adjustment buttons. The time can also be optionally displayed next to the model as well. The time display on the Workspace can be turned on and off within the [Global Layer Settings](#) window. It is helpful to have the time on the Workspace when it is printed.

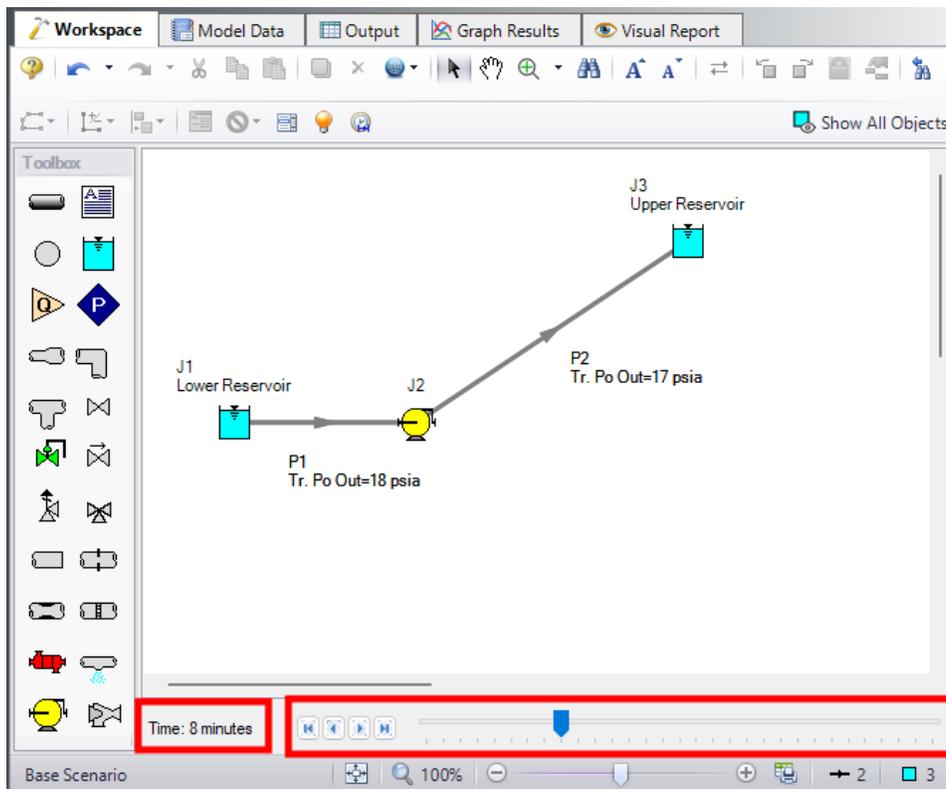


Figure 1: The Workspace has additional time adjustment controls when the XTS Module is in Transient mode

Related Topics

Choosing Time Simulation.....	588
Junction Transient Data.....	588
Finite and Infinite Reservoirs.....	601
XTS in Model Data.....	611
XTS in Output.....	612
XTS in Graph Results.....	613

XTS in Model Data

(XTS Module Only) Junction transient data is displayed on the Transient tab in the Junction section of the Model Data window (Figure 1). This tab is only visible if transient input data exists.

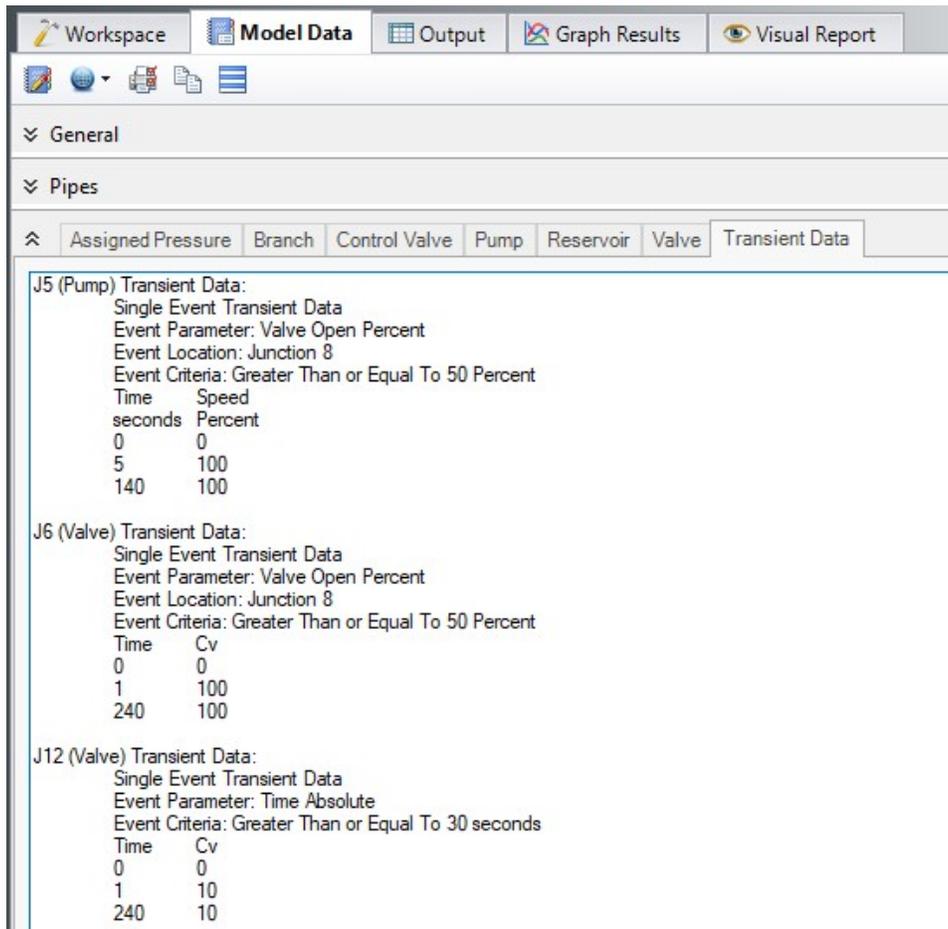


Figure 1: Transient input data is displayed in the Junction section of the Model Data window

Related Topics

Choosing Time Simulation.....	588
Junction Transient Data.....	588

Finite and Infinite Reservoirs.....601
 XTS in Output..... 612
 XTS in Graph Results..... 613
 XTS in Workspace.....610

XTS in Output

(XTS Module Only) Transient simulations create more output data than steady-state runs. This data is displayed in the [Output window](#) in two general ways.

First, the pipe and junction output tables now reflect output at a specific point in time. In Figure 1 below the displayed results are 10 minutes into the simulation. This can be seen at the bottom left of Figure 1. The forward/reverse buttons and the slider at the bottom allow changing the currently displayed time.

Second, the results for all time steps are displayed together in the tables labeled "Transient". Figure 2 shows the same output as Figure 1, except the transient reservoir, pipe and pipetabs have been selected for display. Here one sees multiple results for each pipe or junction. For example, look at the transient pipe data in the middle tables. While the table holds data for all pipes, one can see here only part of the data (up to 3 minutes) for the first pipe, pipe 1. In the far left column, the first data is the pipe number (#1) and the adjacent number in parentheses is the time for that row (in the units defined for the time step interval, - minutes in this case). The time unit of seconds is displayed in the bottom left, and can be changed in the [Output Control](#) time formatting. The data in Figure 2 shows these changes over each time step. The transient tables are organized with a collapsing/expanding capability for each pipe or junction to facilitate data viewing.

Notice that each of the junction Summary reports at the top of Figure 1 (in the General Section, see Pump, Valve and Reservoir Summary) have a parallel Transient table that goes with it.

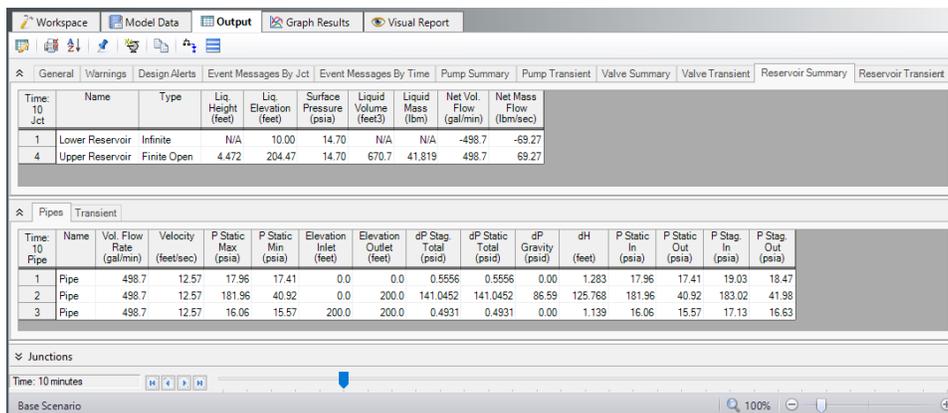


Figure 1: Output window displaying data at 10 minutes

Tank Jct (Time)	Name	Type	Lq. Height (feet)	Lq. Elevation (feet)	Surface Pressure (psia)	Liquid Volume (feet3)	Liquid Mass (lbm)	Net Vol. Flow (gal/min)	Net Mass Flow (lbm/sec)
1 (0)	Lower Reservoir	Infinite	N/A	10.00	14.70	N/A	N/A	-504.0	-70.01
1 (1)	Lower Reservoir	Infinite	N/A	10.00	14.70	N/A	N/A	-503.8	-69.98
1 (2)	Lower Reservoir	Infinite	N/A	10.00	14.70	N/A	N/A	-503.2	-69.90
1 (3)	Lower Reservoir	Infinite	N/A	10.00	14.70	N/A	N/A	-502.6	-69.82
1 (4)	Lower Reservoir	Infinite	N/A	10.00	14.70	N/A	N/A	-502.1	-69.74

Pipe (Time)	Name	Vol. Flow Rate (gal/min)	Velocity (feet/sec)	P. Static Max (psia)	P. Static Min (psia)	Elevation Inlet (feet)	Elevation Outlet (feet)	dP Stag. Total (psia)	dP Static Total (psia)	dP Gravity (psia)	dH (feet)	P. Static In (psia)	P. Static Out (psia)	P. Stag In (psia)	P. Stag Out (psia)
1 (0)	Pipe	504.0	12.70	17.94	17.37	0.0	0.0	0.5673	0.5673	0.00	1.310	17.94	17.37	19.03	18.46
1 (1)	Pipe	503.8	12.70	17.94	17.37	0.0	0.0	0.5667	0.5667	0.00	1.309	17.94	17.37	19.03	18.46
1 (2)	Pipe	503.2	12.68	17.94	17.38	0.0	0.0	0.5655	0.5655	0.00	1.306	17.94	17.38	19.03	18.46
1 (3)	Pipe	502.6	12.67	17.95	17.38	0.0	0.0	0.5643	0.5643	0.00	1.303	17.95	17.38	19.03	18.46
1 (4)	Pipe	502.1	12.65	17.95	17.39	0.0	0.0	0.5631	0.5631	0.00	1.300	17.95	17.39	19.03	18.46

Figure 2: Output window displaying transient data for entire simulation

Related Examples

Beginner - Filling a Tank - XTS	1038
Valve Closing - XTS	1058
Housing Project - XTS	1072
Variable Demand - XTS	1081
Controlled Heat Exchanger Temperature - XTS	1066

Related Topics

Choosing Time Simulation	588
Junction Transient Data	588
Finite and Infinite Reservoirs	601
XTS in Model Data	611
XTS in Graph Results	613
XTS in Workspace	610

XTS in Graph Results

(XTS Module Only) Transient data can graphed in the [Graph Results](#) window in two general ways: Profile graphs and transient graphs.

Pump vs. System curves can still be generated, but only when the Time Simulation is set to Steady Only. See the bottom of this topic.

Profile Graphs

Profile graphs show results where the distance along the pipe is the independent, x parameter (Figure 1). If the pipes are connected, a sequence of pipes can be graphed in the profile. Profile graphs can only show data at a particular time or show overall maximum and minimum values for all times (Figure 2).

Profile graphs can be created for one or more pipe sequences. Note that the x-axis on profile plots is cumulative distance along the pipe sequence.

Multiple profiles can be plotted simultaneously using the [Groups](#) on the Edit menu.

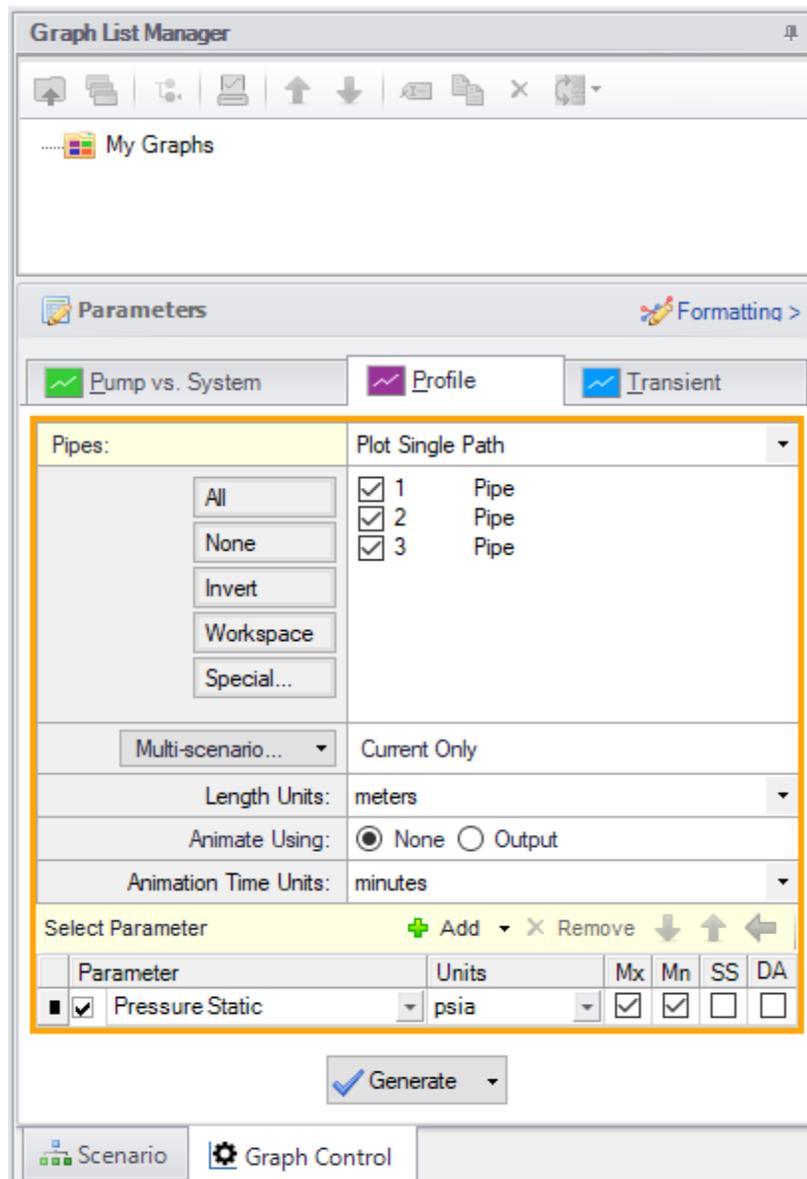


Figure 1: The Graphing panel showing a profile plot for a pipe sequence for maximum and minimum static pressure is to be generated

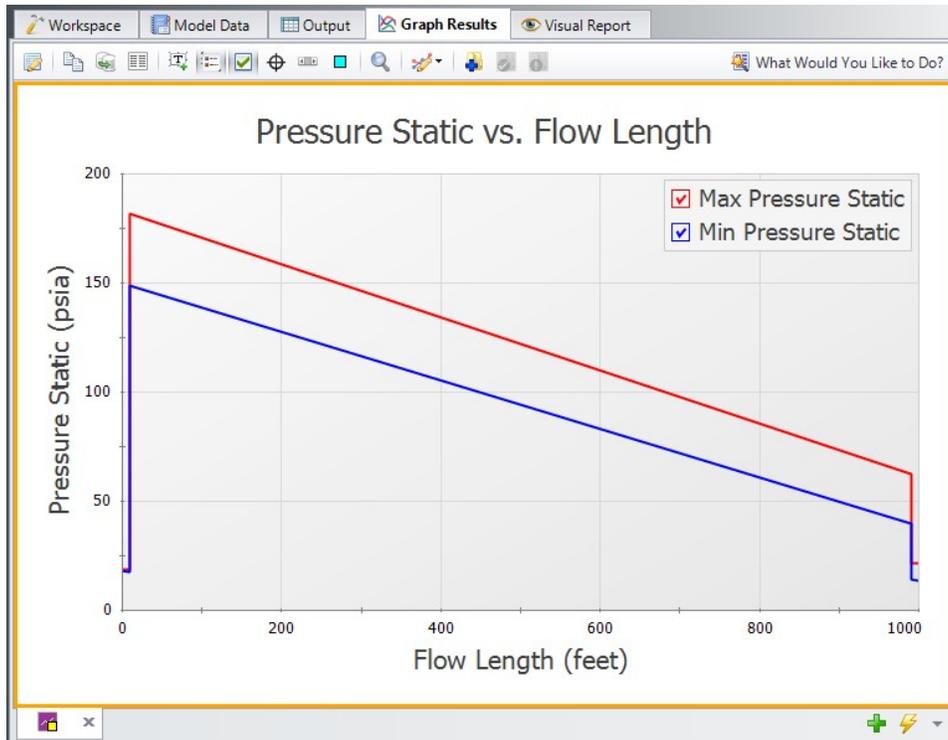


Figure 2: Transient max/min profile plot for pressure as specified in Figure 1

Animation of Output

Another useful feature of profile graphs is that they allow animation of the output results. Animation is selected in the Quick Access Panel (Figure 1) and, when selected, reveals additional control features on the Graph Results window. See [Animating Transient Results](#) for more information.

Transient Graphs

Transient graphs show how parameters vary with time (see Figures 3 and 4). Here the x-axis is time (rather than distance as in profile graphs) as shown in Figure 2.

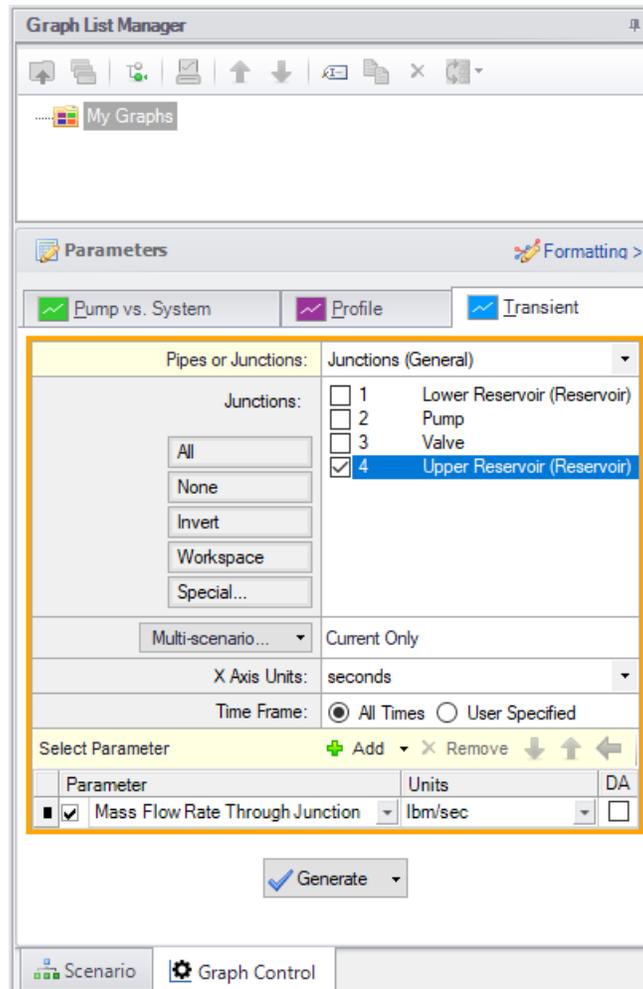


Figure 3: Setting up a Transient graph

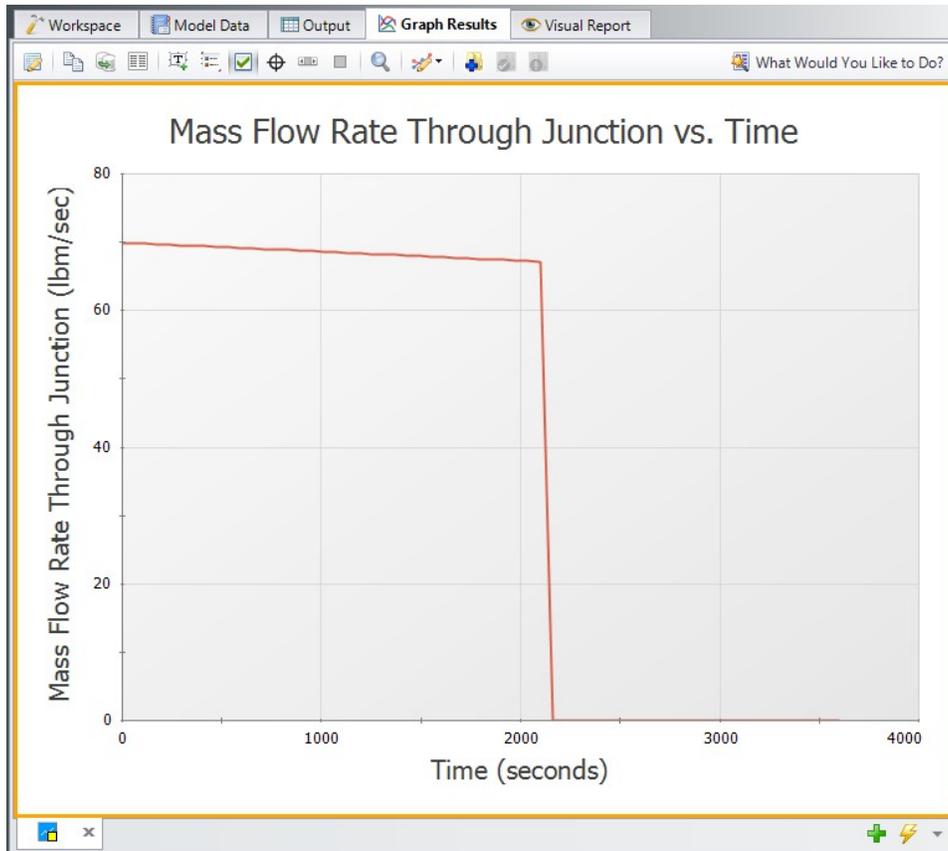


Figure 4: Transient graphs show how parameters vary with time

Pump vs. System Curves

Pump vs. System Curve plots can only be created for a steady analysis. To perform a steady analysis change the Time Simulation to "Steady Only" from the Analysis menu.

To create a pump vs. system curve for a time step other than the initial time step, create a child scenario, change the Time Simulation to "Steady Only," and adjust the model file inputs to reflect the time step you want to create a pump vs. system curve for.

Related Topics

XTS Animating Results.....	618
XTS Quick Graph.....	619
Choosing Time Simulation.....	588
Junction Transient Data.....	588
Finite and Infinite Reservoirs.....	601
XTS in Model Data.....	611
XTS in Output.....	612

XTS in Workspace.....610

Related Examples

Reservoir Balance - XTS 1050
 Variable Demand - XTS 1081

XTS Animating Results

Figure 1 shows the [Graph Results window](#) when using animation. As shown, additional controls become visible to allow the user to adjust animation. At the left is the Play button, which starts the animation. Next to it are the Pause and Stop buttons. If the Pause button is pushed, the animation stops at that time step. One can print the graph or copy the data if desired or use the time slider controls (to the right of the Stop button) to set the time forward or backwards. The Stop button stops the transient and resets the time to zero.

The current time (in seconds) and time step are shown to the right of the time controls. At the far right are the speed controls. Slowing the animation involves adding time delays to the data display. Increasing the speed involves skipping time steps. The default speed, right in the middle, has no delays or skips.

If plotting a single path using the list of pipes in Graph Control (Figure 2), one can cross-plot on the animation design alerts (in Profile Along a Flow Path) or steady-state results (in EGL, HGL and Elevation Profile).

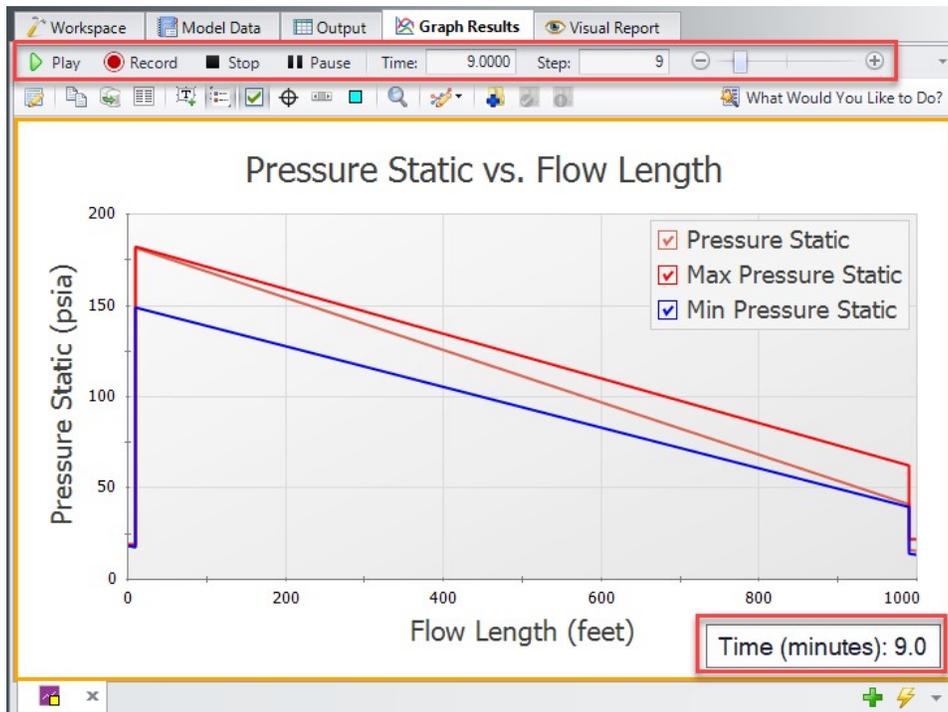


Figure 1: The animation options display additional controls

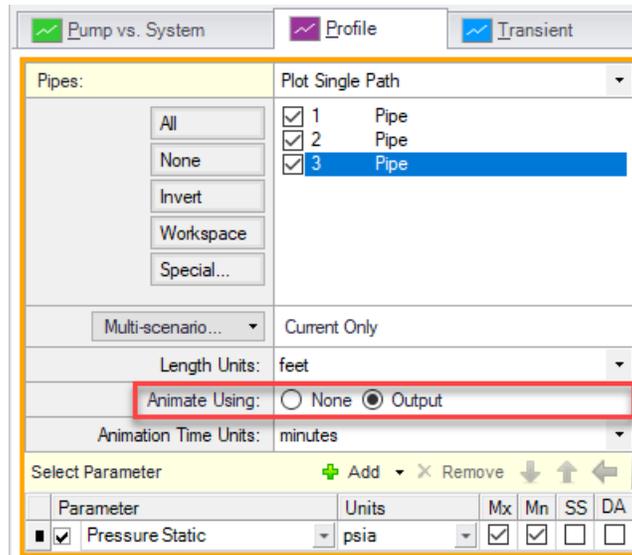


Figure 2: Enabling Animation in Graph Control

Related Topics

XTS in Graph Results.....613
 XTS Quick Graph.....619

XTS Quick Graph

(XTS Module Only) [Transient output data](#) displayed in the [Output window](#) can be graphed by right-clicking on the data in the table and selecting Quick Graph (see Figures 1 and 2).

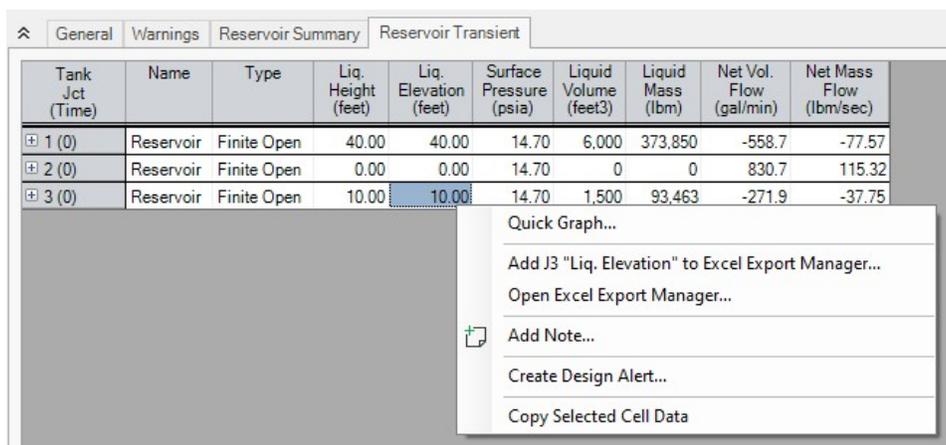


Figure 1: Quick Graph on the Output window is selected by clicking the with the right mouse button in a column of transient data

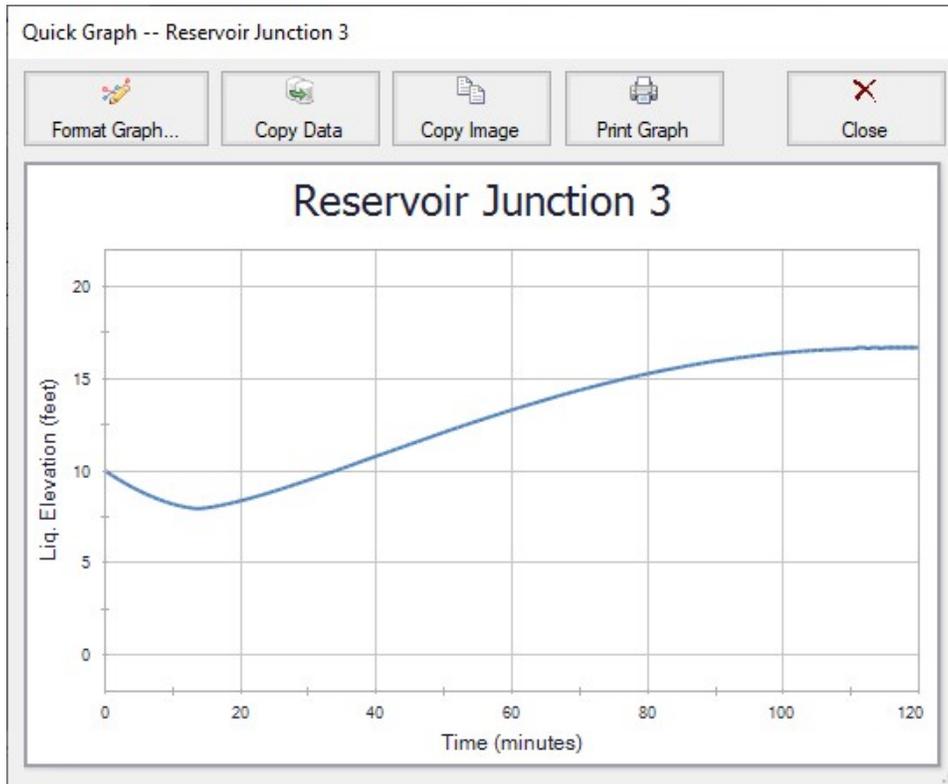


Figure 2: Quick Graph window is opened from Output window as shown in Figure 1

Related Topics

XTS in Graph Results.....	613
XTS Animating Results.....	618

Related Examples

Beginner - Filling a Tank - XTS	1038
---------------------------------------	------

ANS - Automated Network Sizing

What is the Automated Network Sizing (ANS) module?

Sizing a single pipe or a single pump for minimum cost is relatively straightforward with manual iteration. When considering a network of pipes, the manual approach of selecting cheaper components one at a time quickly becomes not only impractical, but ineffective. While manual component selection can certainly *reduce* cost, it often does not result in a *minimized* cost.

The path to *minimizing* cost is often unintuitive. Replacing certain components with more expensive ones often results in a lower total system cost. Solving such a problem - with dozens or perhaps hundreds of variables - cannot reasonably be completed by hand. A computational approach quickly finds ways to reduce cost that may never be found by other means.

In addition, components cannot be arbitrarily changed without impacting overall operation. Any given system has certain requirements that must be met. Reducing pipe sizes too much, for example, may result in flow rates that are unacceptably low. There is a balance between reducing cost and providing the desired operation. The ANS module automates this process, minimizing the cost of complex systems while still meeting requirements with ease.

Note: ANS is not designed to be a **cost estimator**. It instead offers an approach for comparing potential designs using cost as a common denominator. The value of ANS is in the *minimization* of cost, not the precise estimation of total cost.

Using the ANS Module

The ANS module can be enabled from the [Modules panel](#) in [Analysis Setup](#). Activating ANS adds the Automated Sizing item to the Analysis menu, and enabling ANS affects many areas of the AFT Fathom interface.

If the ANS module is active, models can still be run without ANS by selecting Analysis -> Automated Sizing -> Not Used. It is also possible to run models without sizing by selecting Do Not Size in the [Sizing Method panel](#). These options create three distinct possibilities:

- **ANS is not active** - this is standard AFT Fathom mode.
- **ANS is active, but not sizing** - the [Sizing Window](#) and Sizing panels are visible but will not be used in simulation.
- **ANS is active and sizing** - the Sizing Window and Sizing panels are visible and the settings are applied during simulation.

Examples

ANS is best understood by sizing a model yourself. We have created several examples to help walk you through what is required with ANS: [ANS Examples](#).

Related Topics

Modules Panel.....	85
Add-on Modules.....	538
Automatic Sizing Group.....	173
Sizing Summary Panel.....	174
Sizing Fundamentals.....	622
Sizing Window.....	636
Solution Progress with ANS.....	659

Sizing Results.....	660
Workspace Integration.....	662
Junction Loss Factors.....	664
Improving and Troubleshooting the Sizing Process.....	664

Related Examples

Beginner - Three Reservoir Problem - ANS	1134
Control Valve - ANS	1149
Cooling System - Creating Libraries - ANS	1157
Cooling System - ANS	1176
Cooling System, Multiple Design Cases - ANS	1207
Hot Water System - ANS	1217
Housing Project - ANS	1246
Variable Drive - ANS	1231

Sizing Fundamentals

How Does the ANS Module Work?

ANS is built to allow you to approach the automated sizing process in a way appropriate to your system.

You must specify an **Objective - minimizing or maximizing a specific system variable**. Most commonly, the ultimate Objective is to minimize monetary cost.

The pipes to be sized must be defined - any given pipe may be included or excluded from the automated sizing process. It may not be desired to automatically size every pipe in the entire system independently. It is common that groups of pipes - such as those in a header - must be of equal size. This is accomplished by placing them into **Common Size Groups - sets of pipes that are automatically sized but must all be the same size**.

The selection of piping is generally restricted to certain materials, sizes, and types independent of fluid flow concerns. Perhaps certain system piping must be Stainless Steel of any common size, whereas other piping in the system can be regular Steel, but is limited to three on-hand diameters. These are represented with **Candidate Sets - selections of pipe sizes and types that are allowed**.

If the only goal is to minimize cost, the answer is trivial - select the smallest pipe sizes available. Clearly, this is not practical as systems are not built for the sake of being inexpensive, but are meant to accomplish some goal. **These goals, such as minimum flows or pressures, are Design Requirements**. Any number of Design Requirements can be specified in a system, controlling the allowed fluid behavior.

There are several different **Sizing Methods** available. These methods approach the same Objective in different ways, with certain methods being more appropriate for certain systems. Agreement between methods is a strong indication that the ideal sizes have been selected.

Analysis vs. Design

Sizing a system to minimize or maximize a system-wide quantity like monetary cost or pipe weight is a complex process with many considerations. AFT Fathom is foremost an analysis tool, but the ANS module is a design tool. While closely related, analysis and design are different in many ways.

Engineering analysis is the process of using accepted calculation methods to predict the behavior of a given system. Analysis requires the full specification of the system before calculations are made. For example, fundamental fluid equations can be used to determine the flow in a pipe of a known size.

When a new system is being designed, the pipe size is not known. Instead, it might be desired to find the size necessary to meet a minimum flow. The engineer can make an educated guess, perform the analysis, modify the size, and repeat the process. The analytical methods are used iteratively to arrive at a final design.

Even for a simple problem, the iterative *Specify Model -> Analyze -> Investigate Results -> Adjust Model* method can be very inefficient. For simple systems, it is often possible to solve for a design variable by manipulating the fundamental equations. This would allow the process to be completed in one design iteration.

For complex problems direct solutions are impractical or impossible. Instead, the iterative method can be applied with numerical methods. AFT Fathom already automates the "*Specify Model -> Analyze*" portion of the process. AFT Fathom ANS further simplifies the process by using intelligent searching algorithms to automate the "*Investigate Results -> Adjust Model*" steps.

By providing ANS with information on the requirements of a system, it offers a true design-oriented approach, automating a complex iterative process.

Related Topics

Objective.....	623
Design Requirement.....	625
Feasible and Infeasible Designs.....	627
Active, Inactive, and Violated Design Requirements.....	629
Discrete and Continuous Sizing.....	631
Multiple Operating Cases.....	632
Common Size Group.....	633
Initial Costs and Recurring Costs.....	634
Candidate Set.....	636

Related Examples

Beginner - Three Reservoir Problem - ANS.....	1134
---	------

Objective

The **Objective** is the goal of the sizing process - the variable of concern and whether it should be minimized or maximized.

A valid Objective variable must be calculable as a total for any potential system configuration. For example, a system can be sized to minimize or maximize weight because any pipe has a weight measured in a common unit. However, different pipes may have different pressure ratings - it does not make sense to maximize or minimize pressure rating as they cannot be added together and retain the same meaning. Further, the value must be available for all possible configurations - if some pipe sizes are of unknown weight, the sizing cannot proceed effectively because it is unknown whether choosing that size is an improvement or not.

The Objective is defined on the [Sizing Objective panel](#).

Conceptual Example

Consider a simple problem - moving fluid from one large tank to another. The pressures and fluid levels in the tanks are fixed. The fluid must be transferred in a pipe of specialized material, and it is difficult to manufacture this material in small diameters, making them more expensive. The Objective is to select the cheapest pipe possible for the task.

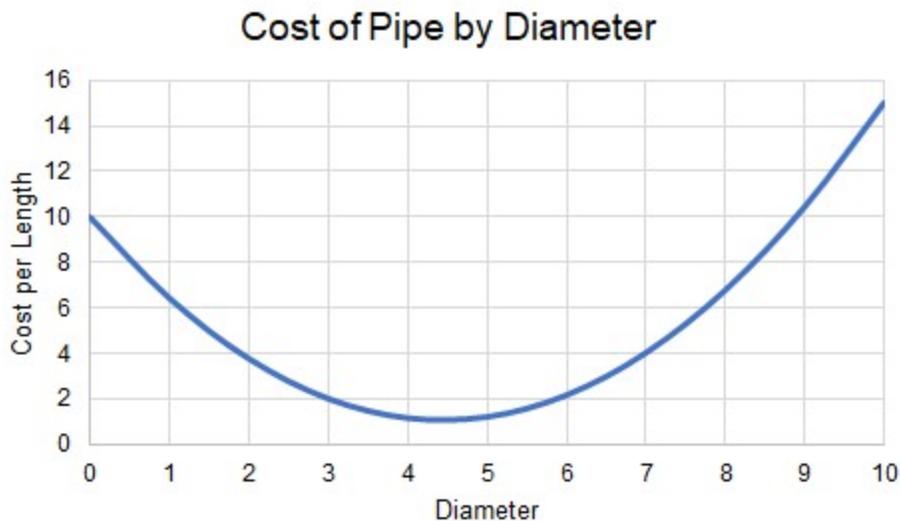


Figure 1: Cost for different diameters of specialized piping

Clearly, there is a minimum cost between diameters "4" and "5."

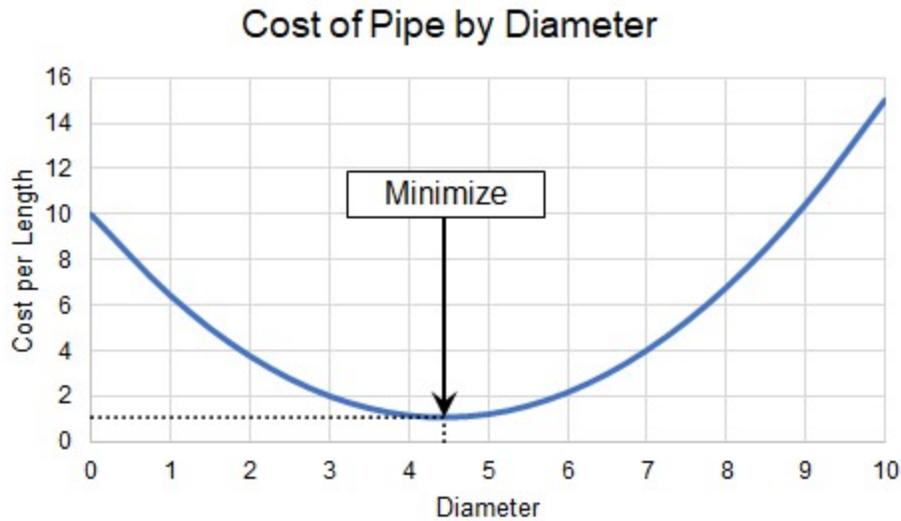


Figure 2: Locating the Diameter with minimum cost

However, what if we are limited by other requirements? For example, we may have a minimum flow rate. This is a [Design Requirement](#), and makes the problem more complex.

Related Topics

Sizing Fundamentals.....	622
Design Requirement.....	625
Feasible and Infeasible Designs.....	627
Active, Inactive, and Violated Design Requirements.....	629
Discrete and Continuous Sizing.....	631
Multiple Operating Cases.....	632
Common Size Group.....	633
Initial Costs and Recurring Costs.....	634
Candidate Set.....	636

Design Requirement

A real system has **Design Requirements - specific parameters that must be met, regardless of the Objective value**. Targeting an Objective without any other associated requirements is not meaningful. If a system is not attempting to achieve some goal, then the minimum cost is zero as no system is required.

The Objective alone may be "How can I drive my cost to a minimum?" but the question really on the designer's mind is "How can I drive my cost to a minimum while maintaining desired operation?"

Design Requirements are defined on the [Design Requirement panel](#).

Note: There is a distinction between a boundary and a Design Requirement. Boundaries represent a *physically forced* flow or pressure while Design Requirements represent *desired behavior*. See [Boundaries vs. Design Requirements](#) for further discussion.

Conceptual Example

Continuing with [our example](#) from the Objective topic, we want to minimize the cost of some pipe. However, there is a minimum flow rate. This is a Design Requirement and limits the possible solutions. In our example, flow rate follows a predictable relationship with diameter (Figure 1).

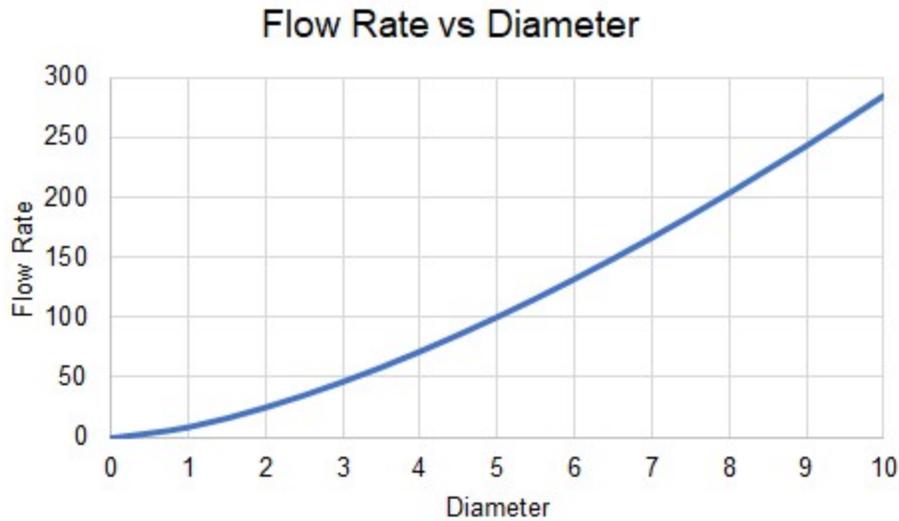


Figure 1: Flow Rates for different diameters, showing the minimum required for our pipe

Our pipe has a Design Requirement that flow rate is at least "150", which indicates a minimum diameter between "6" and "7" (Figure 2).

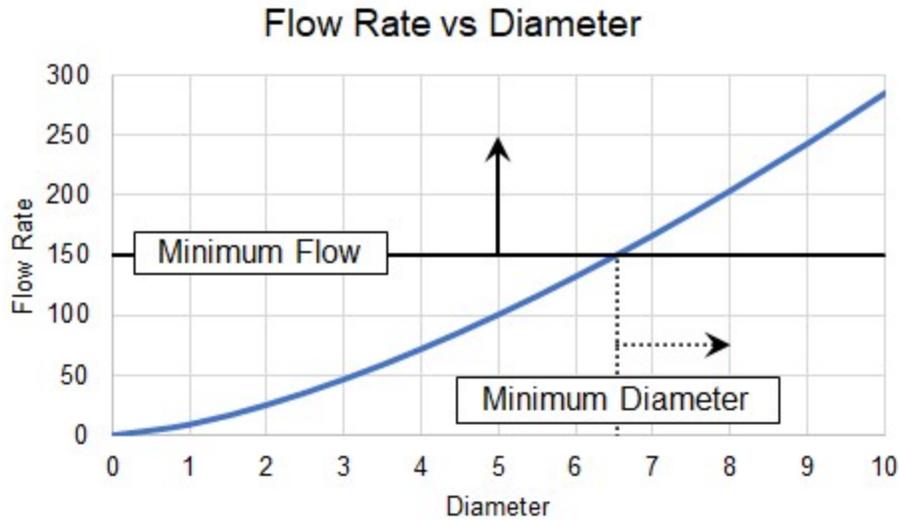


Figure 2: Minimum diameter indirectly imposed by flow rate requirement

Adding Design Requirements may influence the sizing results as this requirement makes certain solutions [Infeasible](#).

Related Topics

Sizing Fundamentals.....	622
Objective.....	623
Feasible and Infeasible Designs.....	627
Active, Inactive, and Violated Design Requirements.....	629
Discrete and Continuous Sizing.....	631
Multiple Operating Cases.....	632
Common Size Group.....	633
Initial Costs and Recurring Costs.....	634
Candidate Set.....	636

Feasible and Infeasible Designs

If a system has Design Requirements, there will be designs unable to meet the requirements. These designs are termed ***Infeasible*** - configurations that violate Design Requirements. Likewise, configurations that meet all Design Requirements are ***Feasible***.

Feasibility can be determined without any knowledge of the Objective. While it is not directly related to the Objective, feasibility can heavily influence the sizing process. It is typical that the ideal solutions are infeasible because they do not meet the Design Requirements. Instead, the best solution is where the Objective is met in the feasible space. Not all Design Requirements will impact the sizing in the same way.

It is important not to confuse *infeasible design* with *impossible design*. A potential design being *infeasible* does not mean that the system could not be built as specified. In fact, it is possible that the system could run quite well. It **only** means that a specified Design Requirement has been violated.

Conceptual Example

We previously [added a Design Requirement](#) to our pipe, which makes certain designs Infeasible (Figure 1). The minimum flow rate Design Requirement requires a minimum diameter.

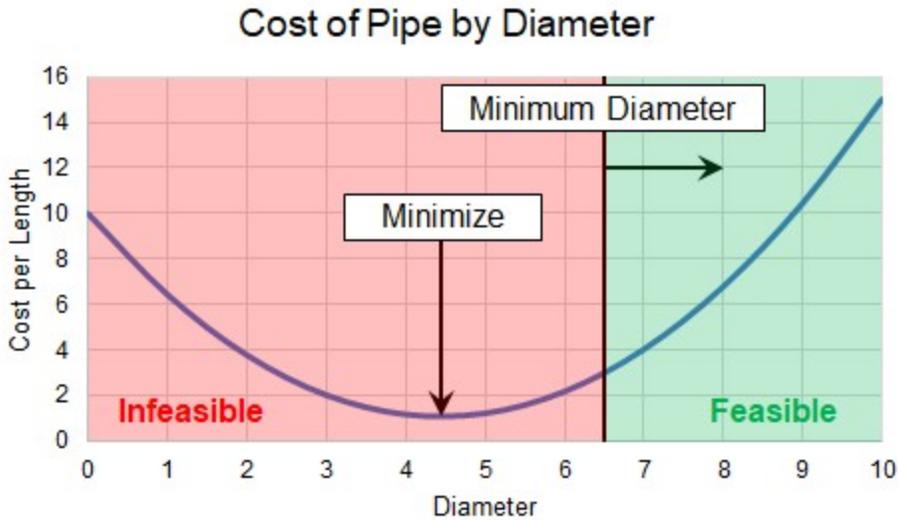


Figure 1: Certain diameters are infeasible when a Design Requirement is added

Clearly, our best case minimum cost is now in the Infeasible region. This would still be the cheapest pipe to select, but would limit flow too much. Instead, we need to find the minimum cost *within the feasible region* (Figure 2).

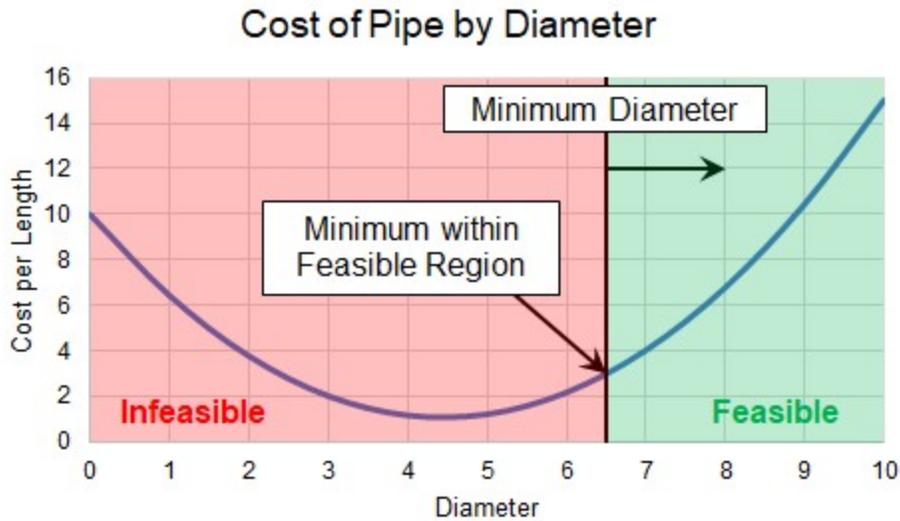


Figure 2: Minimum Feasible Cost

Because our cost only increases with diameters larger than our minimum, the best case cost is now the minimum diameter possible due to the flow requirement. This means that the Design Requirement is [Active](#).

Related Topics

Sizing Fundamentals.....	622
Objective.....	623
Design Requirement.....	625
Active, Inactive, and Violated Design Requirements.....	629
Discrete and Continuous Sizing.....	631
Multiple Operating Cases.....	632
Common Size Group.....	633
Initial Costs and Recurring Costs.....	634
Candidate Set.....	636

Active, Inactive, and Violated Design Requirements

Design Requirements may or may not drive the sizing solution. Design Requirements are *Active* if they drive the sizing solution. Design Requirements that have no impact on the sizing solution are *Inactive*.

There can easily be Design Requirements that, while important for normal operation, do not drive the sizing solution because violating them would not improve the Objective. These Inactive Design Requirements could be removed entirely without changing the sizing solution.

It is not always easy to tell what Design Requirements will be Active or Inactive. A better overall solution can easily often be obtained by making a local selection *more* expensive. This means that Design Requirements that would appear to only affect high-cost options may in fact be Active.

Knowing which Design Requirements are Active or Inactive in a sizing analysis can provide valuable insight into a design. Perhaps one Design Requirement is significantly increasing the cost of the system, but is not critical to the operation. In such cases it may be worthwhile to relax the restriction and modify the Design Requirement for a cheaper solution.

A Design Requirement is Violated if it is not met. During the sizing process it is common that Design Requirements will be Violated. However, if a solution exists, they should only be Active or Inactive in the final result.

Conceptual Example

Adding a specific Design Requirement to our pipe [changed the sizing solution](#). What if we had also added a maximum flow rate, effectively imposing a maximum diameter? It will not impact the sizing solution, despite it being an important consideration for our system. Our original Design Requirement is considered Active, because it is driving the sizing solution, while the second Design Requirement is considered Inactive. As before, the absolute minimum cost diameter would Violate our minimum flow requirement, making it infeasible (Figure 1).

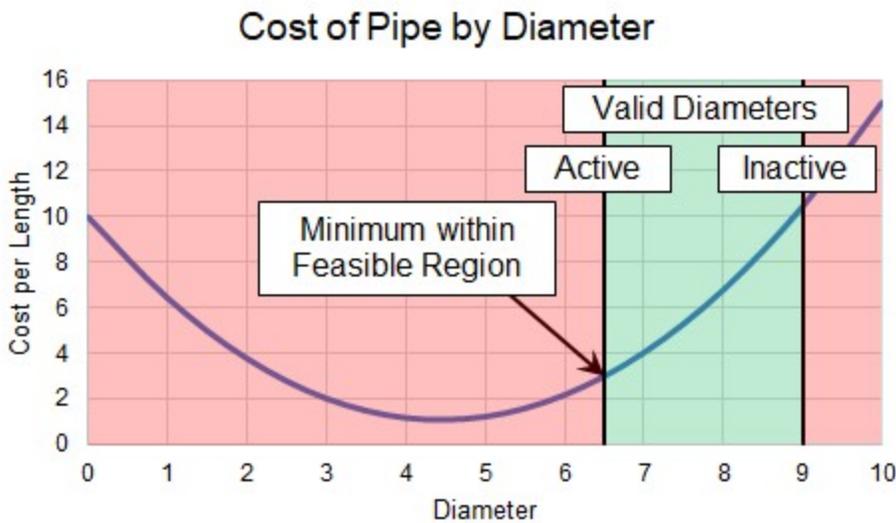


Figure 1: Multiple Design Requirements

Piping is generally not available in any requested size, but instead only in selected, [Discrete](#) diameters, which will also affect the sizing solution.

Related Topics

Sizing Fundamentals.....	622
Objective.....	623
Design Requirement.....	625
Feasible and Infeasible Designs.....	627

Discrete and Continuous Sizing	631
Multiple Operating Cases	632
Common Size Group	633
Initial Costs and Recurring Costs	634
Candidate Set	636

Discrete and Continuous Sizing

Continuous sizing considers any physical diameter in a given range. The ideal solution of a Continuous sizing will contain fractional sizes. However, piping is generally only available in **Discrete sizes - a selection of individual, unique diameters.**

It is assumed during a Continuous sizing that the Objective value varies linearly between intermediate sizes. For monetary cost, this is highly unlikely to be the case - custom pipe of unusual diameter would be very expensive. Continuous sizing is useful because it is generally a *far* easier problem to mathematically solve. On its own, it will give the engineer a good idea of what the ideal system and best-case Objective value looks like. Coupled with a Discrete sizing method, the Continuous sizing result provides an excellent starting point, making the mathematical solution easier.

Continuous sizing methods also generally require less input as they are less restrictive. Starting the sizing process on a Continuous basis is often a good idea as it can reveal potential problems or areas of improvement earlier in the design process.

The selection between Continuous or Discrete is defined on the [Sizing Method panel](#).

Conceptual Example

[Our example](#) has so far been a Continuous sizing problem - we are able to select any size of pipe to drive the cost to an absolute minimum. However, the piping is only available in Discrete sizes. This fact will impact the sizing solution and the resulting cost (Figure 1).

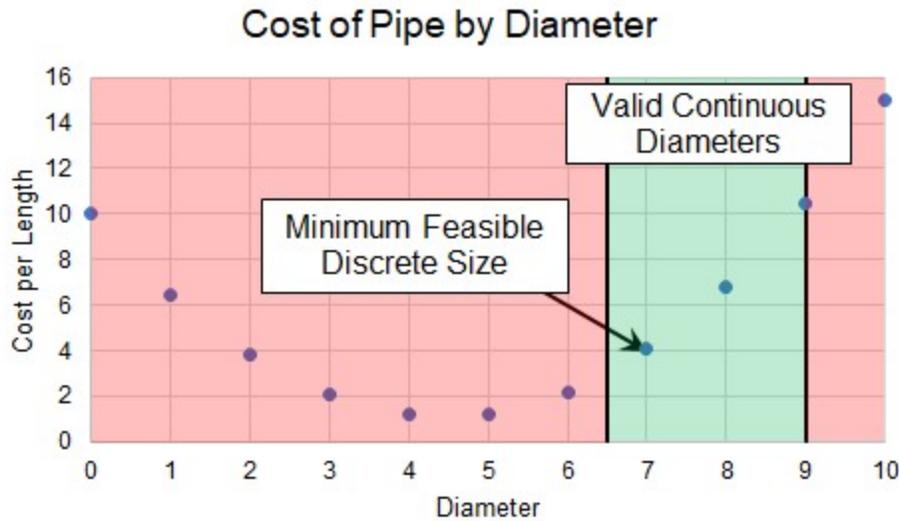


Figure 1: Minimum Feasible Discrete Size

The minimum flow Design Requirement is still be considered Active in this case, as it is still driving the sizing solution.

Note: In our example, the cost of diameter "7" is almost double the cost of diameter "6." In such a case, it may warrant further investigation into why the minimum flow rate is required. If the minimum flow was imposed as a rule of thumb or for a non-critical reason, it is well worth considering violating this Design Requirement and selecting size "6" for the cost savings.

Related Topics

Sizing Fundamentals.....	622
Objective.....	623
Design Requirement.....	625
Feasible and Infeasible Designs.....	627
Active, Inactive, and Violated Design Requirements.....	629
Multiple Operating Cases.....	632
Common Size Group.....	633
Initial Costs and Recurring Costs.....	634
Candidate Set.....	636

Multiple Operating Cases

Sizing a system has many affects in addition to saving on quantities like weight or monetary cost. The operating *capabilities* of the system are impacted by any sizing decision. What may work well for one situation may not work well for another situation. Furthermore, the ideal design for a system that must operate in more than one way may not simply be the ideal design of one of the cases.

In other words, repeatedly sizing the same system for individual multiple operating cases may not result in the ideal design.

By sizing multiple operating cases in parallel, it is possible to account for these variations and still obtain an ideal design in just one sizing simulation. We call these operating cases *Dependent Design Cases*.

This can be enabled on the [Sizing Objective panel](#), with the [Dependent Design Cases panel](#) showing details on the process and model configuration.

Conceptual Example

[Minimizing the size of a pipe](#) with a Design Requirement of a minimum flow is relatively straightforward when only one situation must be considered.

However, consider if our example had multiple operating, or **Dependent Design Cases**. For example, the pressures in the tanks may vary depending on the time of day, the required flow rate may vary with demand, and the environment may vary which could affect the system through thermal affects.

The minimum pipe size found without considering these situations **may violate a Design Requirement** in other cases. Clearly, this cannot be allowed. While some attempt could be made by the engineer to determine the "most severe" case and size for that case, this is often very difficult and can easily neglect potential savings. Instead, by leveraging Dependent Designs, we can account for these variations directly in the model.

Related Topics

Sizing Fundamentals.....	622
Objective.....	623
Design Requirement.....	625
Feasible and Infeasible Designs.....	627
Active, Inactive, and Violated Design Requirements.....	629
Discrete and Continuous Sizing.....	631
Common Size Group.....	633
Initial Costs and Recurring Costs.....	634
Candidate Set.....	636

Related Examples

Cooling System, Multiple Design Cases - ANS.....	1207
Hot Water System - ANS.....	1217
Housing Project - ANS.....	1246

Common Size Group

A **Common Size Group** is a group of pipes that must be of uniform size.

There are two reasons to create Common Size Groups:

- **Physical Requirements** - All of the pipes in the group must be the same size in the field. For example, a long header may be drawn on the Workspace as a series of individual pipe objects. Adding these individual pipes to a Common Size Group forces the entire header to be a common size. If the pipes were not placed in a Common Size Group, it is likely that different sizes would be selected for different sections of the header. Requirements based on pipe connections are also common - it would not be unusual to require equal diameter piping upstream and downstream of a valve or other junction.
- **Reduction in Complexity** - Every additional independently sized pipe increases the difficulty of the sizing problem. Sizing problems with large numbers of independently sized pipes can take extremely long to solve, often for relatively little benefit. This is especially true when the resulting pipe sizes are similar, or when the cost of a particular pipe has minimal impact on the overall cost. For example, parallel pipes would often be expected to have the same size, and short connector pipes will likely have no significant bearing on overall cost if there are much longer pipes being sized. By adding these types of pipes to a Common Size Group, the solution time can be dramatically reduced.

Common Size Groups are defined in the [Size/Cost Assignments panel](#).

Related Topics

Sizing Fundamentals.....	622
Objective.....	623
Design Requirement.....	625
Feasible and Infeasible Designs.....	627
Active, Inactive, and Violated Design Requirements.....	629
Discrete and Continuous Sizing.....	631
Multiple Operating Cases.....	632
Initial Costs and Recurring Costs.....	634
Candidate Set.....	636

Related Examples

Control Valve - ANS.....	1149
Housing Project - ANS.....	1246
Variable Drive - ANS.....	1231

Initial Costs and Recurring Costs

Systems can be sized for a variety of costs. Most notably, there are **Initial Costs** and **Recurring Costs**. The sum of the Initial Costs and Recurring Costs over the expected system lifetime is known as **Life Cycle Cost**.

Initial Costs include any **one-time** cost that is required to get the system in question operational. Most often, this is a combination of Material and Installation costs.

Recurring Costs include any cost that accrues over time (Figure 1). This could be fixed costs at regular intervals, such as the cost of routine Maintenance. It can also be the potentially variable cost of energy to run pumps.

Note: Costs must be defined in [Cost Libraries](#), which are available as part of the base Application.

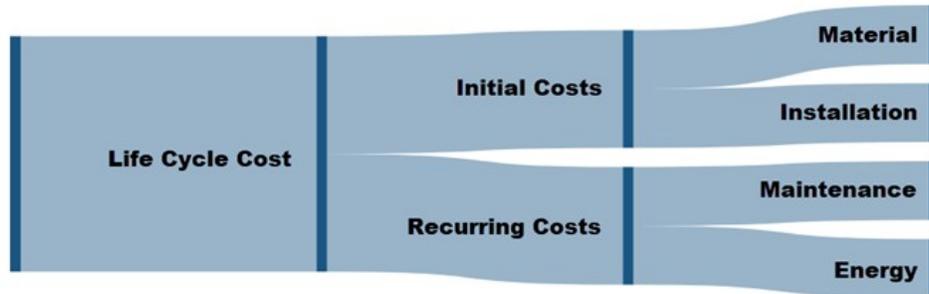


Figure 1: Components of Life Cycle Cost

Keep in mind when considering what costs to include what your most significant cost is likely to be. Systems are often designed to run for many years. This generally means that the total cost of running the system ends up as much more than the cost of building it. Fully defining the sizing input for minimizing Life Cycle Cost is more difficult, but often worthwhile. The inclusion of cost components is defined in the [Sizing Objective panel](#).

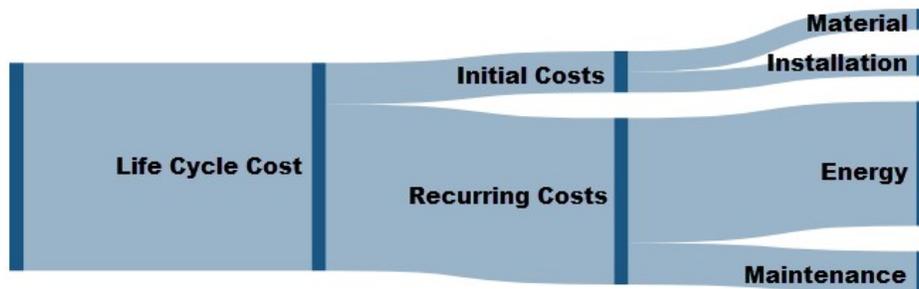


Figure 2: Potential distribution of costs for a long System Life

Related Topics

Sizing Fundamentals.....	622
Objective.....	623
Design Requirement.....	625
Feasible and Infeasible Designs.....	627
Active, Inactive, and Violated Design Requirements.....	629
Discrete and Continuous Sizing.....	631

Multiple Operating Cases.....	632
Common Size Group.....	633
Candidate Set.....	636

Related Examples

Cooling System - ANS.....	1176
---------------------------	------

Candidate Set

A **Candidate Set** is a set of allowed pipe sizes. All pipes that are being sized must be associated with a Candidate Set.

The Candidate Set is required for both [Discrete and Continuous Sizing](#). In continuous sizing it is used to determine a minimum and maximum pipe size. The intermediate sizes are only used if sizing is done on a discrete basis.

Multiple Candidate Sets can exist, but only one can be applied to any given pipe or [Common Size group](#).

Candidate Sets, and their assignment to pipes, are defined on the [Candidate Sets panel](#).

Related Topics

Sizing Fundamentals.....	622
Objective.....	623
Design Requirement.....	625
Feasible and Infeasible Designs.....	627
Active, Inactive, and Violated Design Requirements.....	629
Discrete and Continuous Sizing.....	631
Multiple Operating Cases.....	632
Common Size Group.....	633
Initial Costs and Recurring Costs.....	634

Related Examples

Beginner - Three Reservoir Problem - ANS.....	1134
---	------

Sizing Window

The Sizing Window is a sixth [Primary Window](#) that appears when the [ANS module is activated and enabled](#) (Figure 1).

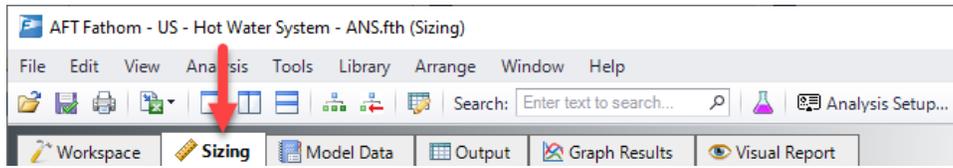


Figure 1: The Sizing Window

The Sizing window contains all of the information required to define a sizing analysis. Other areas of AFT Fathom interact with the definition of sizing parameters, but ultimately all changes are reflected here. The Sizing window is further divided into Sizing panels (Figure 2).



Figure 2: Sizing Navigation

Each Sizing panel serves a certain purpose in the sizing process.

- [Sizing Objective panel](#) - What parameter should be minimized or maximized?
- [Size/Cost Assignments panel](#) - What pipes or junctions should be sized, which should be counted in the cost total, and how should they be grouped together?
- [Candidate Sets panel](#) - What materials, sizes, and types of pipes can be considered in the sizing process?
- [Design Requirements panel](#) - What requirements does the system have?
- [Assign Cost Libraries panel](#) - If optimizing on cost, what is the source of the cost data?
- [Sizing Method panel](#) - What type of sizing and what method should be used?
- [Dependent Design Cases panel](#) - What operational variations must the sizing analysis account for?
- [Sizing Summary panel](#) - How are the sizing settings currently applied to the system?

Panel Locks

Every Sizing panel can be individually locked to prevent accidental changes (Figure 3).

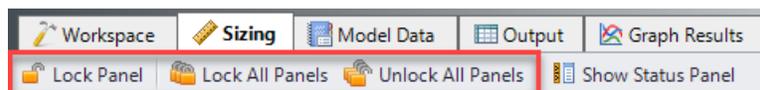


Figure 3: Locking panels

Related Topics

Sizing Objective Panel.....	638
Size/Cost Assignments Panel.....	642
Candidate Sets Panel.....	646
Design Requirement Panel.....	648
Assign Cost Libraries Panel.....	650
Sizing Method Panel.....	652
Dependent Design Cases Panel.....	653
Sizing Summary Panel.....	656

Related Examples

Beginner - Three Reservoir Problem - ANS	1134
--	------

Sizing Objective Panel

Sizing Options



Figure 1: The three sizing ANS sizing methodologies are shown

- **Perform Sizing** - The best case selection of pipe sizes will be determined based on the various Sizing Window settings.
- **Calculate Cost, Do Not Size** - Pipe sizes will not change from their initial, user-defined size. The costs of the system as-defined will be calculated.
- **Do Not Size** - No sizing or cost calculations will be carried out. This is equivalent to temporarily disabling ANS.

Objective

The [Objective](#) is the goal of the automated sizing process. Any of the listed parameters can be minimized or maximized.

- **Pipe Weight** - Calculated using the pipe material density and dimensions from the internal pipe material library for the pipes being sized.
- **Pipe and Fluid Weight** - Calculated using the pipe material density and dimensions from the internal pipe material library, as well as the fluid density defined on the [Fluid panel](#) of [Analysis Setup](#).
- **Flow Volume** - The total volume of fluid in the pipes calculated from the inner diameter and geometry defined in the Pipe Properties window.
- **Pipe Inner Surface Area** - The internal surface area of the pipes calculated using the inner diameter and geometry defined in the Pipe Properties window.
- **Monetary Cost** - The monetary value of the pipes/junctions calculated from the user provided [cost libraries](#)

Minimizing a physical parameter such as weight, volume, or surface area is often a good approximation of minimizing Initial Cost. These options are also useful in specific cases where monetary cost of the fluid system is not the primary concern.

Other Options

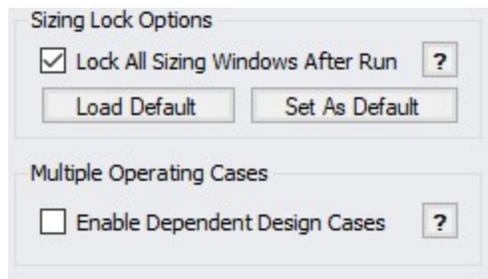


Figure 2: The additional options in the Sizing window are shown

By default, all sizing panels are locked after a simulation is run. Modifying sizing input will require unlocking the panel.

It is possible to model [multiple operating cases](#) simultaneously by enabling Dependent Design Cases (Figure 2). The [Dependent Design Cases panel](#) is hidden unless it is enabled here.

Monetary Cost Options

There are several considerations to be made when considering the total Monetary Cost of a piping network. Is the Initial Cost of the Materials required the only concern? Do Installation Costs have to be accounted for? The true cost of a system could be considered its [Life Cycle Cost](#) - how much does it cost not only to build the system, but to run and maintain it? These are all valid approaches to the challenge of minimizing cost, and each requires different input.

Size for Initial Cost			
Size for Initial Cost, Show Energy Costs			
Size for Initial and Recurring Costs			
User Specified			
Cost Type	Include in Sizing Objective	Cost Report Only	Ignore Cost
Initial Costs			
Material	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>
Installation	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>
Recurring Costs			
Energy	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>
Maintenance	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>

Figure 3: The various life cycle costs that can be considered are shown

- **Size for Initial Cost** - Material and Installation Costs will be used to size the system.
- **Size for Initial Cost, Show Energy Costs** - Material and Installation Costs will be used to size the system. Energy cost does not impact the sizing, but will be displayed. This may be helpful to determine if it is worthwhile to resize the system considering Recurring Costs.
- **Size for Initial and Recurring Costs** - Energy Cost is considered in the sizing, along with Material and Installation Cost.
- **User Specified** - Custom sizings are possible by selecting the appropriate radio buttons. For example, if Installation Costs are not readily available, these can be ignored by changing that row to Ignore Cost.

Note: The Sizing Level affects the available options. If the overall Sizing Level is "Do Not Size" then no sizing can take place and this will be reflected by forcing all cost rows to Ignore Cost.

- **Warn When Fittings & Losses Costs are Missing** - By default, ANS will warn the user whenever costs are missing for any [Fitting & Loss](#) defined within a sized pipe. For the most accurate solution, these costs should be included, however, they are often small compared to other costs and this option allows ignoring those warnings.

Recurring Cost Options

If any Recurring Costs is specified to be included in the Cost Report or Sizing, additional input is required (Figure 4).

The screenshot shows a configuration window for cost calculation. It is divided into three sections:

- System Life (For Recurring Costs):** Contains three input fields: 'System Life' set to 20 with a 'years' dropdown, 'Interest Rate' set to 15 with a '%' dropdown, and 'Inflation Rate' set to 2 with a '%' dropdown and a 'Per years' dropdown.
- Energy Cost:** Contains two radio buttons: 'Use Energy Cost Libraries' (unselected) and 'Use This Energy Cost Information' (selected). Below are input fields for 'Cost' (0.03) with a 'U.S. Dollars' dropdown, and 'Per' (kW-hr) with a dropdown.
- Initial Cost Limit:** Contains a checkbox 'Initial Cost Limit' (unchecked) and a 'Max Initial Cost' field with a 'U.S. Dollars' dropdown.

Below the 'Energy Cost' section, there is a blue note: "Only this fixed energy cost set here will be used. All Energy Cost settings in the Library Manager will be ignored."

Figure 4: Additional required information when recurring costs are included

- **System Life (For Recurring Costs)** - The period for which the Recurring Costs are counted. Interest and Inflation Rates can optionally be specified.
- **Energy Cost** - Only enabled if Energy is included in the Cost Report or Sizing.
 - **Use Energy Cost Libraries** - Use [Cost Libraries](#) for energy cost.
 - **Use This Energy Cost Information** - Specify a monetary cost for energy directly. If this is specified, Energy Cost Libraries will not be utilized.
- **Initial Cost Limit** - Limits the total amount allowed for Initial Cost. A higher initial cost may result in a lower Life Cycle cost, but it is sometimes the case that there is an absolute limit on initial spending.

Related Topics

Sizing Window.....	636
Size/Cost Assignments Panel.....	642
Candidate Sets Panel.....	646
Design Requirement Panel.....	648
Assign Cost Libraries Panel.....	650
Sizing Method Panel.....	652
Dependent Design Cases Panel.....	653
Sizing Summary Panel.....	656

Related Examples

Beginner - Three Reservoir Problem - ANS	1134
--	------

Size/Cost Assignments Panel

The Size/Cost Assignments panel defines which components are to be included or excluded from sizing and cost reporting. Buttons across the top of the panel display selections for a specific component type.

Pipes

The options for pipes include including or excluding them from sizing, including or excluding from cost, including in cost only if size changes, and collecting the pipes into [Common Size Groups](#) (Figure 1).

- **Sizing and Cost Options** - This section of the table controls sizing and cost calculation.
 - **Automatically Size** - If either of these columns are selected, this pipe will be included in the sizing analysis.
 - **Always Include in Cost** - The cost of this pipe will always be considered and reported.
 - **Include in Cost if Size Changes** - The cost of this pipe will effectively be considered zero for the initial size. This is useful for representing existing pipes, while considering the cost savings that could result from replacing the existing pipe with one of a more appropriate size.
 - **Do Not Size** - If either of these columns are selected, this pipe will **not** be included in the sizing analysis.
 - **Include in Cost** - The pipe will not be considered for sizing, but the cost of the pipe for the user specified size will be included. This is useful for having a complete picture of the system cost, even if certain pipes must be a given size.
 - **Do Not Include in Cost** - The pipe will not be sized or included in costs.
- **Pipe Grouping** - Pipes can be placed into Common Size Groups here. You can sort this table by clicking on the column headers, or by right-clicking on the headers and choosing Organize Grid by Common Size Group.
 - **Not in Group** - the pipe is not in a group, and is either not sized, or sized independently from any other pipe.
 - **Common Size Groups** - This section is only visible after a Common Size Group is created with the New button. When a Common Size Group is created, it will create a corresponding column underneath the Common Size Groups header. Pipes can then be placed into the Common Size Group by selecting the radio button for the given pipe under the given Common Size Group. Groups can be collected to the top of the table by clicking on their column headers.

Cost Calculation

Pipes	Sizing and Cost Options				Pipe Grouping					
	Automatically Size		Do Not Size		Not in Group	Common Size Groups				
	Always Include in Cost	Include in Cost if Size Changes	Include in Cost	Do Not Include in Cost		Main Lines	Pump Station Small	Pump Station Large	HEX A	HEX B
P1	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
P2	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
P3	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
P4	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
P5	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>

Figure 1: The options for including pipes in both sizing and cost calculation. Additionally, the options to create custom size groups is shown.

Note: You can also add Pipes to Common Size Groups [directly from the Workspace](#), and optionally color the Workspace pipes to indicate which pipes are in what groups.

Pipes can be selected by clicking on the row header, and multiple pipes can be selected by holding control or shift. With multiple pipes selected, their sizing or cost options can be reassigned with the buttons near the top of the table. All of the tables displayed in this panel include the [selection features](#) (Figure 2) available elsewhere in AFT Fathom.

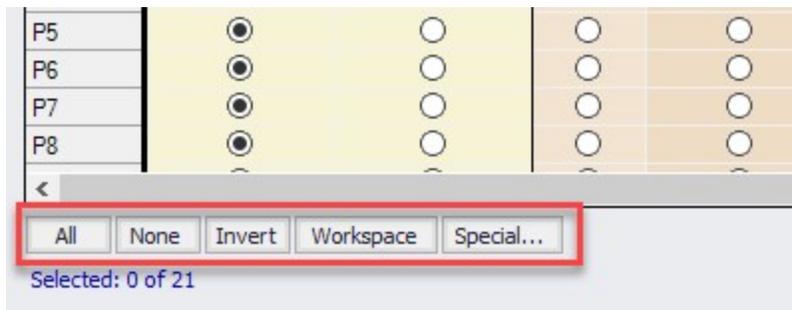


Figure 2: Selection options available for the Size/Cost Assignments panel

Selected rows can have their settings changed all at once with the buttons just above the table (Figure 3).

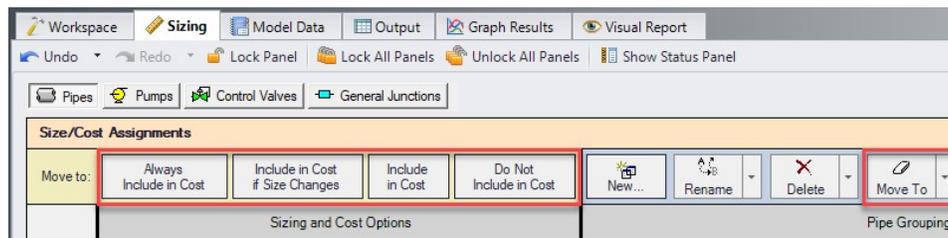


Figure 3: The buttons above the table can be used to change selected rows all at once

Note the indication of the total number of Independent Pipe Sizes in the lower right (Figure 4). This quantity has important implications on run time and what Search Method will be appropriate for the model.

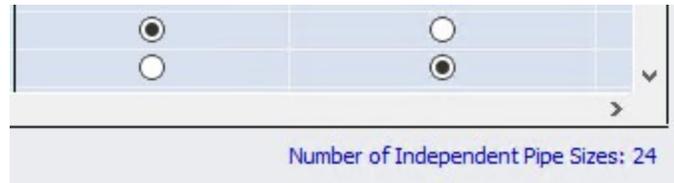


Figure 4: The Number of Independent Pipe Sizes in the model are shown which has implications on runtime

Pumps, Control Valves, and Junctions

- **Cost Options** - This section of the table controls sizing and cost calculation (Figure 5).
 - **Include in Cost Report and Sizing** - If a junction is included in the sizing calculation, this does not mean that the junction itself is modified as pipes are. Instead, this means that the cost associated with the junction is included in the Objective. This can indirectly affect the size of the junction as the pipes are sized accordingly around it.
 - **Include in Cost Report Only** - The junction cost will **not** be included in the Objective. However, the cost of the junction will be included in the Cost Report. The sizing of the junction may change as a result of pipe size changes, but the cost impact of this change will not drive the solution.
 - **Do Not Include Cost** - Not included in sizing or costs.

^ Pumps	Cost Options		
	Include in Cost Report and Sizing	Include in Cost Report Only	Do Not Include Cost
J3 (Pump#1)	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>
J6 (Pump#2)	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>

Figure 5: Cost options available for Pumps, Control Valves, and Junctions

Maximum Cost Groups

Pumps and Control valves can optionally be grouped together in maximum cost groups (Figure 6). This is useful to account for situations where the cost of multiple components is the same, regardless of how the system operates around them.

For example, it is common to have two identical pumps installed in parallel, with only one operating and the other functioning as a spare. Clearly, the cost of purchasing and installing two identical pumps will be identical regardless of how they end up being operated. The automated sizing calculation will base the cost of the pump off its required size to maintain the desired flow. However, in the case of a spare there is no flow requested, meaning that the ideal size of pump is "zero" - no pump at all! ANS must be told that

the spare pump is the same size as the running pump. This will force the costs to be the same, accounting for both in the sizing.

In any Maximum Cost Group, the **largest cost** found during sizing will be applied to **all** of the junctions in the group for the given category. There are three categories:

- **Material** - The most common grouping - A set of identical junctions will likely cost the same to procure, and therefore should be placed into a Material Maximum Cost Group.
- **Installation** - Commonly tied to Material - the junctions are likely to cost the same to install.
- **Maintenance** - May or may not be grouped - if all junctions have routine maintenance whether or not they are used, they should be grouped. If maintenance occurs only when the junction is operating for some amount of time, then a spare will have different maintenance costs and they should not be grouped.

Maximum Cost Groups					
Material		Installation		Maintenance	
Not in Group	VendorA	Not in Group	VendorA	Not in Group	VendorA
<input checked="" type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
<input checked="" type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
<input checked="" type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>

Figure 6: The Maximum Cost Groups feature is shown allowing for items with identical costs to be grouped together

Related Topics

Sizing Window.....	636
Sizing Objective Panel.....	638
Candidate Sets Panel.....	646
Design Requirement Panel.....	648
Assign Cost Libraries Panel.....	650
Sizing Method Panel.....	652
Dependent Design Cases Panel.....	653
Sizing Summary Panel.....	656

Related Examples

Beginner - Three Reservoir Problem - ANS	1134
Housing Project - ANS	1246
Variable Drive - ANS	1231

Candidate Sets Panel

[Candidate Sets](#) are selections of pipe Material, Types, and Sizes that can be considered in the automated sizing process.

Define Candidate Sets

Before it can be applied to any pipes in the model, a Candidate Set must be defined.

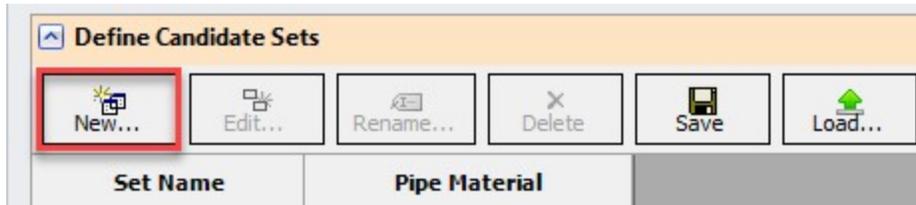


Figure 1: The new button allows for the creation of new Candidate Sets

Clicking New will request a name for the new Candidate Set (Figure 1). After entering a name, the Select Pipe Sizes window will appear. In this window, a Material must be selected, and any number of specific Types and Sizes can be added for consideration (Figure 2). It would be common to select commercially available Sizes of a certain Type, but any selection can be made.

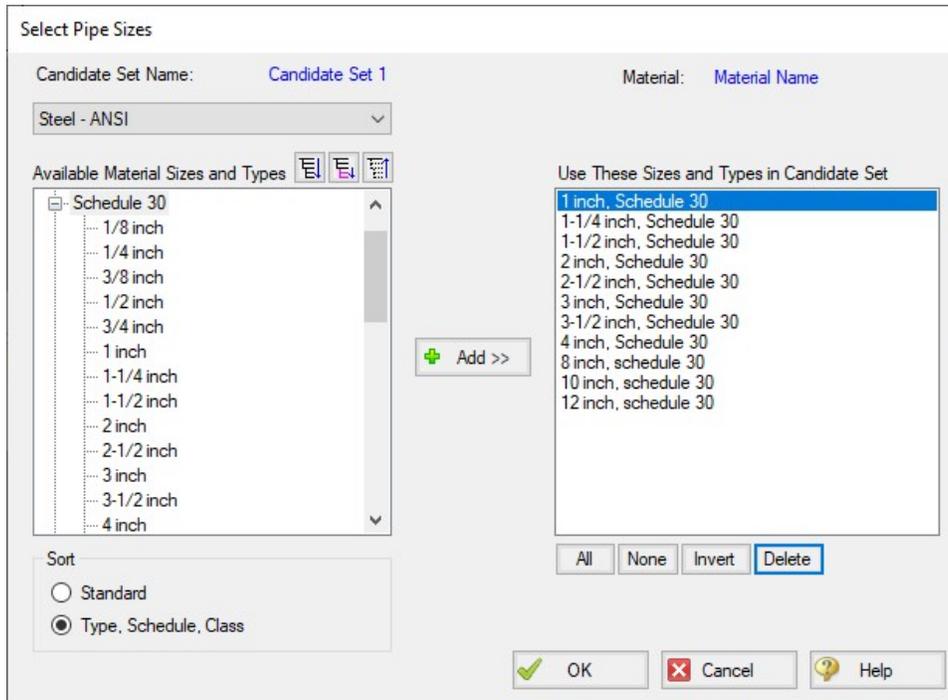


Figure 2: The Select Pipe Sizes window allows for the sizes to be considered for sizing to be specified

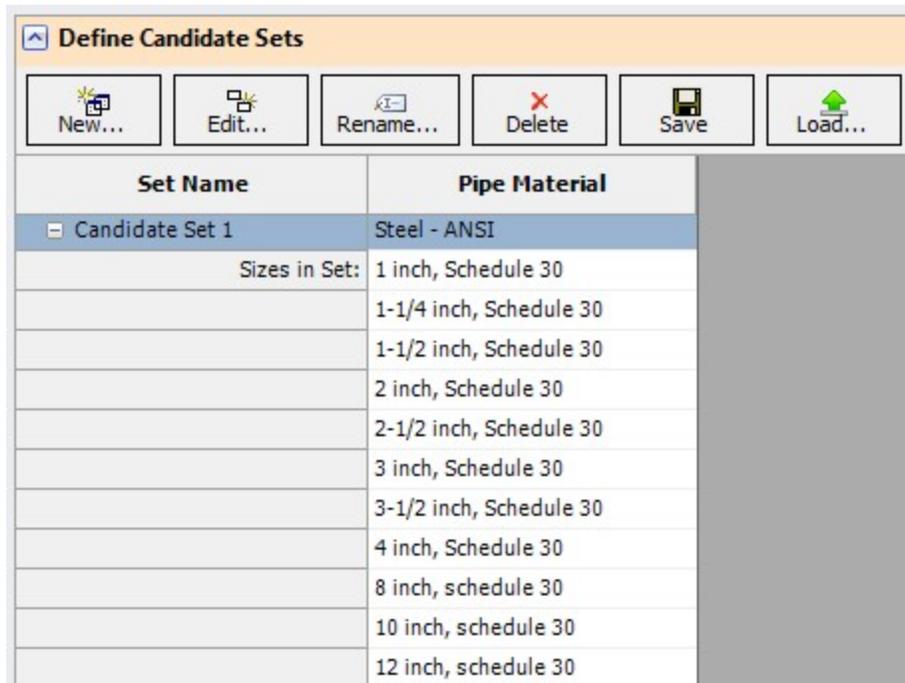


Figure 3: A fully defined candidate set

Candidate Sets can be saved to and loaded from a file for distribution or use in other models.

Note: The Candidate Sets defined here are common to all [scenarios](#). They can be applied or not applied in certain scenarios as needed.

Assign Candidate Sets to Pipes

Every pipe that is being sized must have an associated Candidate Set (Figure 4). As Common Size Groups already define every pipe within them as the same Size, Candidate Sets are assigned to the entire Common Size Group. Multiple Common Size Groups or Not in Group pipes can be assigned at once by selecting the rows and using the Assign To button.

Pipes	Candidate Sets		
<input type="checkbox"/> Common Size Groups	No Set	STD Steel Pipe 1-36 inch	Candidate Set 1
Main Lines	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
Pump Station	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
Pump Station Headers	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>

Figure 4: Size Groups can be assigned Candidate Sets to limit the sizing options

Related Topics

Sizing Window.....	636
Sizing Objective Panel.....	638
Size/Cost Assignments Panel.....	642
Design Requirement Panel.....	648
Assign Cost Libraries Panel.....	650
Sizing Method Panel.....	652
Dependent Design Cases Panel.....	653
Sizing Summary Panel.....	656

Related Examples

Beginner - Three Reservoir Problem - ANS	1134
--	------

Design Requirement Panel

[Design Requirements](#) are of critical importance in a sizing process. Without any Design Requirements, the system will be sized based on cost alone. This will tend to reduce pipe diameters to the minimum size allowed by the [Candidate Set](#). In turn, the fluid behavior of the system may become physically untenable - for example, with extremely high fluid velocities.

Additionally, the very requirements of the system may not be met - certain outflows may require a given pressure and flow and lowering the pipe size too much may mean these needs cannot be met.

Design Requirements ensure that the selected pipe sizes are the minimum cost considering these limitations - the minimum [Feasible](#) cost.

Define Design Requirements

Pipes and Junctions can both have Design Requirements applied to them (Figure 1). In both cases, individual Design Requirements are defined with a Name, Parameter, Location, and limiting Value.

	Name	Parameter	Location	Max/Min	Value	Units
1	Max Velocity	Velocity	N/A	Maximum	12 feet/sec	
2	Min Pressure	Pressure Static	Overall	Minimum	15 psia	
3	Min Clr Flow	Volumetric Flow Rate	N/A	Minimum	1900 gal/min	

Figure 1: Defined Design Requirements in the Define Pipe Design Requirements

Note: The Design Requirements defined here are common to all [scenarios](#). They can be applied or not applied in certain scenarios as needed.

Assign Design Requirements

Created Design Requirements can be applied to any number of applicable objects, and a single object can have any number of Design Requirements applied to it. [Common Size Groups](#) are indicated here, and a Design Requirement can be applied to an entire group with ease (Figure 2). If this is not desired, the group can be expanded and individual selections can be made.

Common Size Groups	Max Velocity (< 12 feet/sec)	Min Pressure (> 15 psia)	Min Clr Flow (> 1900 gal/min)
<input type="checkbox"/> Pressurizer	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
P1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
P22	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/> Pump Station	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/> Clr Piping	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
<input type="checkbox"/> Clr Headers 1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/> Clr Headers 2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/> Not in Group	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>

Figure 2: Created Design Requirements can be assigned to multiple objects

Related Topics

Sizing Window.....	636
Sizing Objective Panel.....	638
Size/Cost Assignments Panel.....	642
Candidate Sets Panel.....	646
Assign Cost Libraries Panel.....	650
Sizing Method Panel.....	652
Dependent Design Cases Panel.....	653
Sizing Summary Panel.....	656

Related Examples

Beginner - Three Reservoir Problem - ANS	1134
--	------

Assign Cost Libraries Panel

If Monetary Cost is selected as the [Objective](#), the costs of any components considered in the Objective must be available.

Note: For an accurate analysis, costs are required for **any component that may be considered in sizing**. This means that it is not sufficient to have cost data only for some pipe sizes, or only for certain sized junctions. Every pipe Size and Type considered in a given [Candidate Set](#) needs to have cost information for every parameter considered in the Objective.

Cost and Energy Libraries

Libraries are not discussed in detail here. Both Cost and Energy Libraries are available as a part of the base application. For a thorough explanation see [Cost Overview](#) and the associated topics. The [Cooling System - Creating Libraries - ANS](#) is an excellent place to start if you are unfamiliar with Cost Libraries.

Assigning Cost Libraries

Candidate Sets can have any number of Cost Libraries assigned to them (Figure 1). This can be useful as different sources of costs can be distributed among several libraries. For example, different Sizes of pipes may be available only from certain manufacturers, and splitting libraries by material source can provide clarity.

Likewise, Installation cost may have several sub-component costs such as Shipment and Labor. In these cases, There would be an Installation cost for the same Material, Size and Type in two different libraries. Including both of these libraries will use the sum of the two costs in the Objective.

Candidate Sets	Cost Library		
	All	Selected	
STD Steel Pipe 1-36 inch	<input type="radio"/>	<input checked="" type="radio"/>	<input checked="" type="checkbox"/>
STD Steel Pipe 36-48 inch	<input type="radio"/>	<input checked="" type="radio"/>	<input checked="" type="checkbox"/>

Figure 1: Candidate sets can have multiple Cost Libraries assigned to them

Note: Keep in mind when assigning cost libraries that **the automated sizing will consider the sum of costs from all selected cost libraries**. Be careful to include multiple libraries only if every cost would be incurred. Do not include, for example, two cost libraries comparing costs from two different suppliers.

Modifying Cost Calculations

Multipliers on a given cost category for a given component can be applied here. These are useful in a variety of situations. A pump may be in a difficult to access location, driving up its maintenance cost by some factor. Or, a control valve may only operate a percentage of the time, meaning that less cost is required for maintenance. A pipe that must be buried would have a higher installation cost. There are many such examples where a specific pipe or junction in the system has a higher or lower cost than the value defined in the library.

Note: Multipliers can also be applied within cost libraries. The multipliers are independent of each other and stack multiplicatively.

If recurring costs are considered, there will be an additional column in this table titled Service Duration. By default, recurring costs are considered for the entire System Life. If certain components should only be considered as incurring cost for part of the System Life, this can be defined here.

Pumps	Multipliers (percent)				Service Duration (years)			
	Material	Installation	Maintenance	Operation	Full	Partial	Start	End
J4 (Pump C)	100	100	100	100	<input checked="" type="radio"/>	<input type="radio"/>	0	20
J7 (Pump B)	100	100	100	100	<input checked="" type="radio"/>	<input type="radio"/>	0	20
J10 (Pump A)	100	100	70	100	<input type="radio"/>	<input checked="" type="radio"/>	0	5

Figure 2: The Multipliers and Service Duration fields allow for modification of the library cost

Related Topics

Sizing Window.....	636
Sizing Objective Panel.....	638

Size/Cost Assignments Panel.....	642
Candidate Sets Panel.....	646
Design Requirement Panel.....	648
Sizing Method Panel.....	652
Dependent Design Cases Panel.....	653
Sizing Summary Panel.....	656

Related Examples

Cooling System - Creating Libraries - ANS	1157
Cooling System - ANS	1176

Sizing Method Panel

The Sizing Method panel controls how the automatic sizing will determine the ideal sizes.

Sizing Level

- **Continuous** - Allow the sized pipes to be any diameter
- **Discrete** - Restrict the sized pipes to diameters directly specified in their Candidate Set

[Discrete and Continuous Sizing](#) each have their own advantages. Discrete Sizing will present a set of pipes that are commercially available, as defined by the associated [Candidate Sets](#). Continuous Sizing is generally much easier computationally, and hence faster.

It is good practice to begin with Continuous Sizing and only proceed to Discrete Sizing when the general design of the system has been determined.

Search Method

There are many ways to approach the search for a minimum or maximum.

A foolproof way to determine if a system is really the "best" is to compare every single possible combination allowed. Then, it will be guaranteed that the lowest cost option has been found. Unfortunately, attempting to [brute force](#) the problem is only viable with a very small number of variables. Instead, the solution must be searched for in an intelligent manner, without explicitly checking every possible option.

There are several such methods available in ANS:

- **Continuous Methods** - These methods use [gradients](#) - rates of change of the Objective (total system cost) with respect to one design variable (a pipe diameter). For the most basic system, the gradient will show an increasing cost with increasing pipe size. However, this quickly becomes less rigid when considering Design Requirements and the impacts a certain pipe size has on the system. A smaller pipe size may make the pipe itself cost less, but it may require a larger pump which costs more, effectively making the gradient show a *decreasing* cost with increasing size. Regardless, if the direction of decreasing cost is known, an intelligent choice to search in that

direction can be made. **By intelligently running sizing sub-problems, it is possible to find a discrete solution with continuous methods.** The [Branch and Bound](#) method searches for a discrete minimum by placing additional requirements on certain variables (branching) and analyzing the results of the continuous solution to determine if this section of the design space is worth further investigation (bounding).

- [Modified Method of Feasible Directions](#) - Reliable and applicable to most problems.
- [Sequential Quadratic Programming](#) - Frequently the most efficient, but more sensitive to system definition.
- [Sequential Unconstrained](#) - Sensitive to initial sizes and control parameters, but more capable of handling large numbers of uniquely sized pipes.
- **Discrete Methods** - Some methods are inherently discrete, and can only consider discrete possibilities during the sizing process. These methods are sometimes more adept at finding global minimums, but are also generally much slower than gradient based methods.
 - [Genetic Algorithm](#) - Cross-breeds a random "population" of designs, mimicking biological evolution. Less likely to become trapped in a local minimum, but very slow compared to other methods.

Note: Full understanding of the Search Methods is not required for effective use of ANS. However, it is important to realize that different methods are better suited to different systems. It is recommended to use multiple Search Methods on any system to ensure the best case sizes are located, and to determine if one method is more efficient than the others.

Related Topics

Sizing Window.....	636
Sizing Objective Panel.....	638
Size/Cost Assignments Panel.....	642
Candidate Sets Panel.....	646
Design Requirement Panel.....	648
Assign Cost Libraries Panel.....	650
Dependent Design Cases Panel.....	653
Sizing Summary Panel.....	656

Related Examples

Beginner - Three Reservoir Problem - ANS	1134
--	------

Dependent Design Cases Panel

Dependent Designs are a powerful way of accounting for different [operating cases](#) during the sizing process. This panel displays information on how to create a Dependent Design Case, or if one exists, how the objects are related to their base objects.

Enabling Dependent Design Cases

Dependent Design Cases are enabled by checking the box on the [Sizing Objective panel](#).



Figure 1: The Enable Dependent Design Cases option selected

Note: Once a Dependent Design Case exists, this option cannot be turned off unless the Dependent Design Case is completely removed.

Creating a Dependent Design Case

Once the option above has been enabled, it is possible to create Dependent Design Cases on the [Work-space](#).

This is done simply with [Duplicate Special](#). Select all of the objects to be included in the Dependent Design and select Duplicate Special from the Edit menu, and selecting the special **Make Dependent Design Case** option.

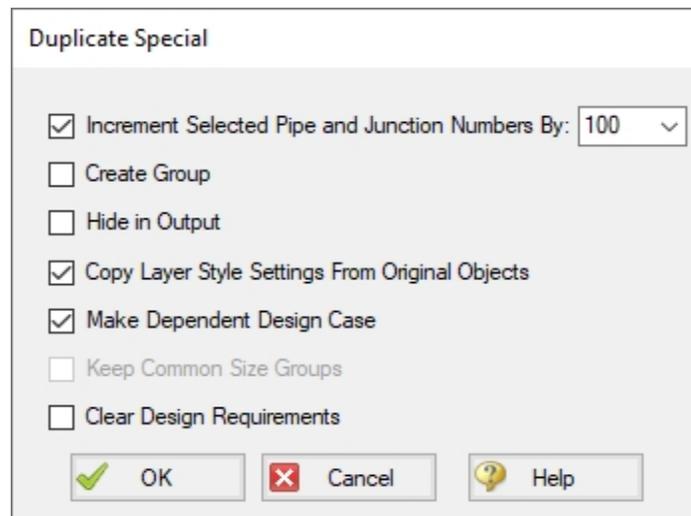


Figure 2: The Duplicate Special window can be used to create Dependent Design Cases

Selecting OK will duplicate all of the objects selected and increment their numbers by the indicated value. In addition, the new objects will be related to their base objects.

Modifying the Dependent Design

Dependent Design objects represent the **same** object that they were created from. Logically, the end result must be that the objects are the same in all dependent designs. It does not make logical sense for pipes to be different lengths, for example. ANS will force a single pipe diameters across all dependent designs.

However, there are many modifications that are reasonable, and they fall into two categories:

- **Modifying the physical configuration of the model** - Environmental changes can affect temperature, changing processes can change the fluid, tank liquid levels can change, pumps may be brought online, valves may change position, and even new pipes may be added to represent a planned expansion. These are just a few examples of a different operating case being the result of a physically changed model.
- **Modifying the [Design Requirements](#)** - The exact same physical system may still have different requirements under different conditions. A plant may have different demands during day or night, or it may be desired to account for future production without modifying the piping layout. These are a couple of examples of changing Design Requirements to create different operating cases.

Note: The only sizing input that can be changed for a Dependent Design are the Design Requirements. Remember that both the base object and Dependent Design object really represent just one physical object - it cannot have different cost or be of a different pipe material in different cases.

Reviewing Dependent Designs

As the model becomes larger and more complex, it may be difficult to tell what objects correspond to others across operating cases. The Dependent Design Cases panel (Figure 2) shows a review of how the objects are connected:

Dependent Design Cases		
The grid below shows the relations of Dependent Design Case (DDC) objects to their base objects, as well as their design requirements. Note that Sizing Option, Common Size Group, Maximum Cost Group, Candidate Set, and Cost Libraries/Modifiers will always be synchronized between base objects and their Dependent Design Case objects. This grid is informational only. To change parameters, go to the necessary Sizing Panel.		
Pipes	DDC Pipe?	Assigned Design Requirements
P1	NO	Max Vel
P101	YES (P1)	Max Vel
P2	NO	Max Vel
P102	YES (P2)	Max Vel

Figure 3: The Dependent Design Cases panel

Related Topics

Sizing Window..... 636

Sizing Objective Panel.....	638
Size/Cost Assignments Panel.....	642
Candidate Sets Panel.....	646
Design Requirement Panel.....	648
Assign Cost Libraries Panel.....	650
Sizing Method Panel.....	652
Sizing Summary Panel.....	656

Related Examples

Cooling System, Multiple Design Cases - ANS.....	1207
Hot Water System - ANS.....	1217
Housing Project - ANS.....	1246

Sizing Summary Panel

The Sizing Summary panel gives an at-a-glance, but comprehensive, view of how the model has been configured for sizing.

All of the sizing input can be grouped into a few relatively simple categories. However, as the model becomes more complex, the relationships between these inputs can be difficult to visualize. For this reason, the Sizing Summary is a relational diagram with a handful of parent nodes. Nodes can be turned on or off with the checkboxes above the diagram.

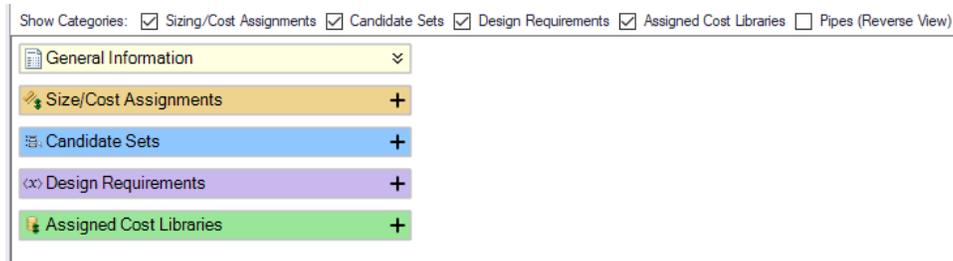


Figure 1: The Sizing Summary panel on launch

Each node can be expanded to see the related entries.

Note: The Summary can be manipulated much like the Workspace - use the scrollwheel to zoom in and out, and pan the view with the scrollbars or by clicking the middle mouse button.

Cost Calculation

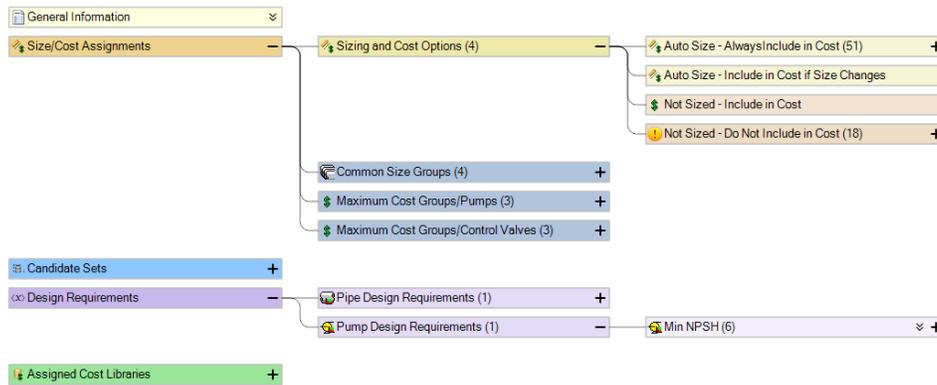


Figure 2: The Sizing Summary panel with options expanded

Individual objects can be expanded, showing detailed related information about them.

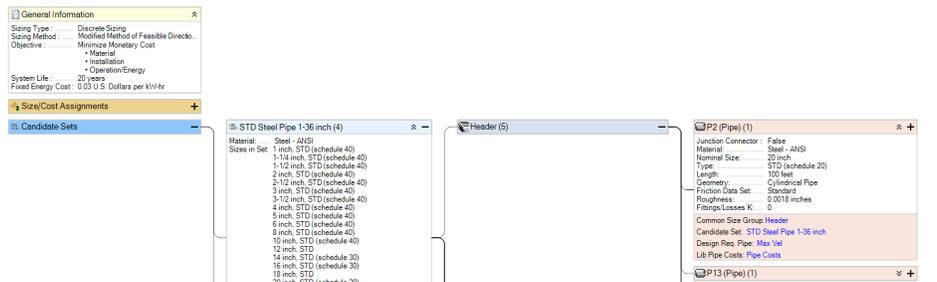


Figure 3: The Sizing Summary panel can show detailed related information

Clicking on a related category for a specific object will indicate the associated node(s).

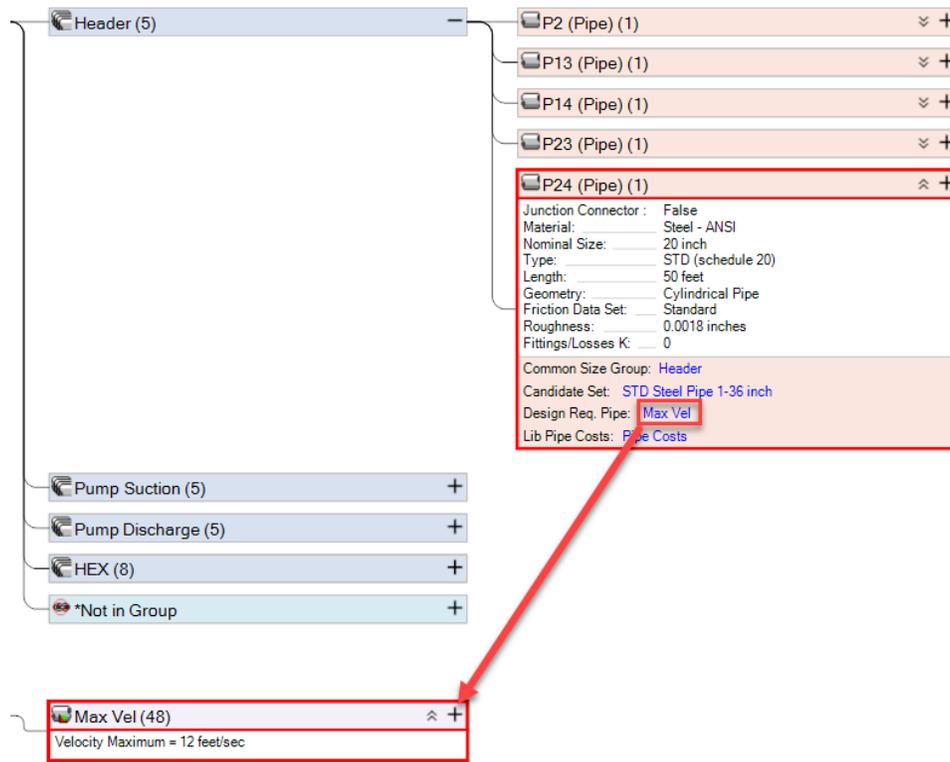


Figure 4: Associated nodes can be determined by clicking on a related category

Related Topics

Sizing Window.....	636
Sizing Objective Panel.....	638
Size/Cost Assignments Panel.....	642
Candidate Sets Panel.....	646
Design Requirement Panel.....	648
Assign Cost Libraries Panel.....	650
Sizing Method Panel.....	652
Dependent Design Cases Panel.....	653

Related Examples

Cooling System - ANS.....	1176
---------------------------	------

Solution Progress with ANS

When a model is run with the ANS module enabled, an additional line will be shown in the Solution Progress window to indicate the progress that the ANS module has made towards reaching the specified goal. There are three fields displayed, as can be seen in Figure 1 below.

The "Best Feasible Cost" and "Current Cost" fields indicate the total cost for the model in the default units, as indicated in the Solution Progress window. The Current Cost field shows the cost calculated for the last design iteration, while the Best Feasible Cost field tracks the cost of the best solution so far that meets all design requirements.

The field on the far right of the Sizing Calls to Solver line tracks the total iterations performed by ANS. For every ANS iteration, ANS makes a call to the hydraulic solver in Fathom to determine the hydraulic solution for the current sizing solution. Thus, this value will always be smaller than the flow and head iterations.

Other Actions

If the solution is paused by clicking the Pause button in the Solution Progress window, there are several options that can be selected from the Other Actions window to assist with convergence.

Selecting Solution Control allows the Solution Control options to be adjusted before calculations are resumed. This is not recommended unless you have a good understanding of the parameters in the [Steady Solution Control Group](#), or this has been suggested by AFT Technical Support.

The Sizing Control option provides a file with detailed information on the sizing parameters being used by the Sizing Engine. This can provide insight for cases where ANS is not converging, or is taking a long time to converge.

The Accept Sizing option will be available from the Other Actions window if the Genetic Algorithm is selected on the [Sizing Method panel](#). This option allows the user to stop the iterations and accept the design with the Best Feasible Cost as the solution for the model. Accepting the sizing solution early is helpful for models which have been running for a long time, without seeing any significant improvement on the Best Feasible Cost for the model. Rather than spending additional time on find the absolute best design, it may be desired to accept the design that gives the current Best Feasible Cost instead to save time.

Cost Calculation

Solution Progress - Paused

Maximum Iterations: 1000000 Run Time: 12.69

Relaxation: Automatic
 Sizing Method: Genetic Algorithm - Start Continuous MMFD - Population = 126
 Sizing Goal: Discrete - Objective Costs Include: Material, Installation, Operation/Energy

	Absolute Tolerance Max Out of Tol.	Relative Tolerance Max Out of Tol.	Total Iterations
Head: 1.0E-04 Relative Change Not used (Absolute Change meters)	1.857E-04	9.888E-08	8964
Vol. Flow Rate: 1.0E-04 Relative Change Not used (Absolute Change m3/hr)	3.989E-03	4.397E-06	1325
Temperature:			

Sizing Calls To Solver: (Objective Cost Units - U.S. Dollars)	Best Feasible Cost	Current Cost	
	5,079,364	5,081,121	660

Setting up input...
 Determining connectivity...
 Checking sizing setup...
 Initializing model...
 Running Solver...
 Initial design is infeasible. Cost of initial design is 33,160,272 (U.S. Dollars) ...
 Performing continuous sizing...
 Continuous cost was... 5,077,241 (U.S. Dollars) (Solver Calls = 237, Time = 4.71) ...
 Performing discrete sizing...

Other Actions ▾
 Continue
 Cancel
 Graph Results...
 Visual Report...
 Output...

Figure 1: The Solution Progress window has an additional line to show information on the ANS Calls to Solver when the Fathom model is run with ANS enabled

Sizing Results

Automatically Selected Pipe Sizes

The pipe sizes corresponding to the best case Objective value are reported in the Output Window.

⌵ Pipes Sizing Pipe Design Requirements Library Sources					
Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (inches)
10	Pipe	Steel - ANSI	12 inch	STD	12.00
11	Pipe	Steel - ANSI	12 inch	STD	12.00
12	Pipe	Steel - ANSI	12 inch	STD	12.00
13	Pipe	Steel - ANSI	20 inch	STD (schedule 20)	19.25
14	Pipe	Steel - ANSI	20 inch	STD (schedule 20)	19.25

Figure 1: The sizing tab of the output window shows the pipe sizes indicated by the ANS module

Cost Report

The Cost Report is displayed in the top section of the Output Window. It will display the appropriate variable for the Objective (e.g. kilograms for a Weight sizing). All Cost Libraries that were used in the generation of the report are listed in Library Sources. It is described in detail in [Cost Report](#).

Design Requirement Status

Whether or not a [Design Requirement is Active](#) can have important bearing on system design decisions. Values in **yellow** are active. Values highlighted in **blue** are Design Requirements that were active during Continuous sizing, and may appear inactive after Discrete sizing. Un-highlighted values are inactive.

Pipe	Velocity Maximum < 8 feet/sec [Max Pipe Velocity]
1	0.000
2	6.614
3	5.643
4	5.643
5	5.643
6	7.045
7	5.736
8	5.736
9	7.045

Jct	% Flow of BEP Minimum > 80 Percent [Min % BEP]	% Flow of BEP Maximum < 100 Percent [Max % BEP]
4	94.26	94.26
7	95.81	95.81
10	94.26	94.26

Figure 2: Active Design Requirements in the Output window are indicated by blue highlighting

Changing Input Sizes to Resulting Pipe Sizes

- **Set Pipes to Resulting Pipe Sizes** - The [Transfer Results to Initial Guesses](#) takes on additional functionality when ANS is enabled. After a sizing run, you can use this feature to transfer the resulting pipe sizes to the pipes. For certain Search Methods, this can greatly speed up the solution process for future sizing runs.
- **Generating a Disconnected Scenario** - Generating a [disconnected scenario](#) allows the sizing results to be saved as input in a new scenario. This can be useful to keep track of results from different methods, or to tweak the sized solution if required.

Output Control

Items included in the Cost Report can be changed by adding/removing items from the Cost Report tab on the Display Parameters tab of Output Control. Similarly, the items shown in the Sizing tab can be controlled from the Pipe Sizing tab.

Workspace Integration

Modifying Sizing from the Workspace

Right-clicking anywhere on the Workspace allows the selection of existing [Common Size Groups](#):

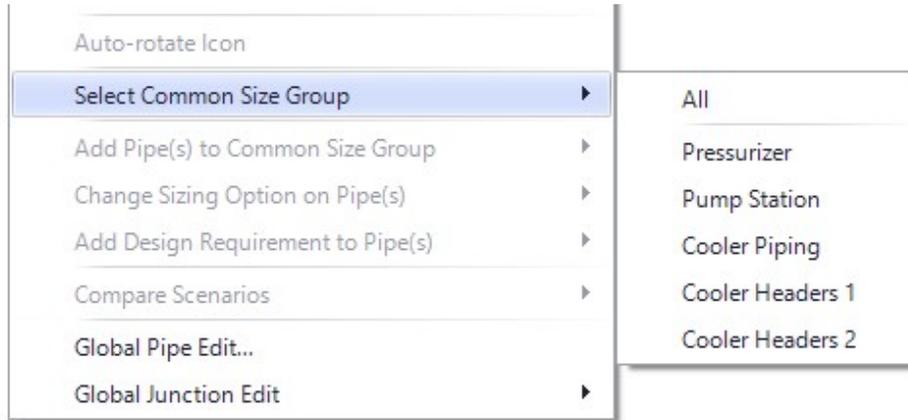


Figure 1: Objects can be added to Common Size Groups from the right-click menu

Selected pipes can be moved to a new or existing Common Size Group:

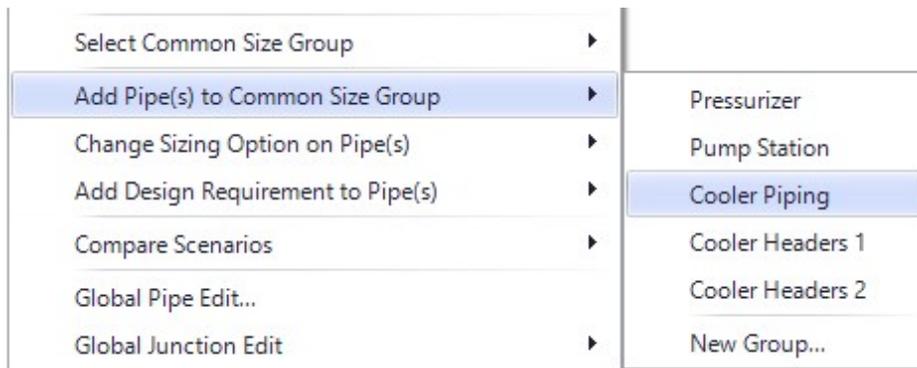


Figure 2: Pipes can be added to a previously created Common Size Group

Changes will be previewed before they are applied:

The following changes will be made to Common Size Groups:

Pipe Number	Old Group	New Group
9	Pump Station	Cooler Piping

Figure 3: Preview of changes to Common Size Groups

Sizing Level [Assignments](#) and [Design Requirements](#) can also be applied in this manner.

Viewing Assigned Common Size Groups and Design Requirements

Additionally, each Common Size Group or Design Requirement can have a unique color applied to it by selecting the appropriate option under the View menu.

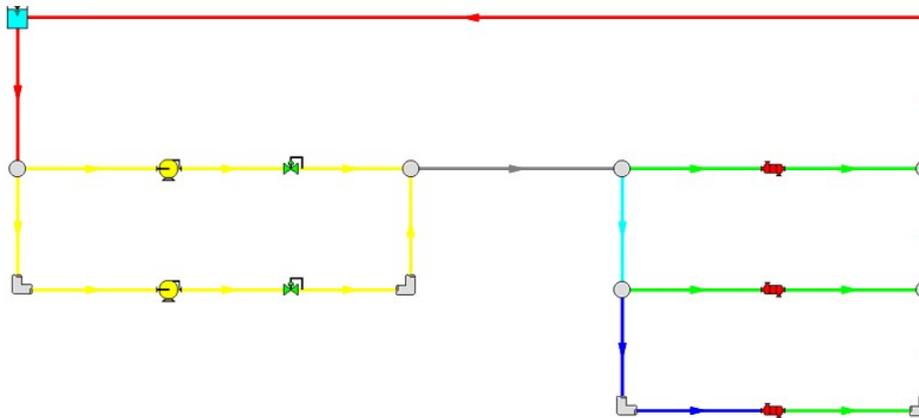
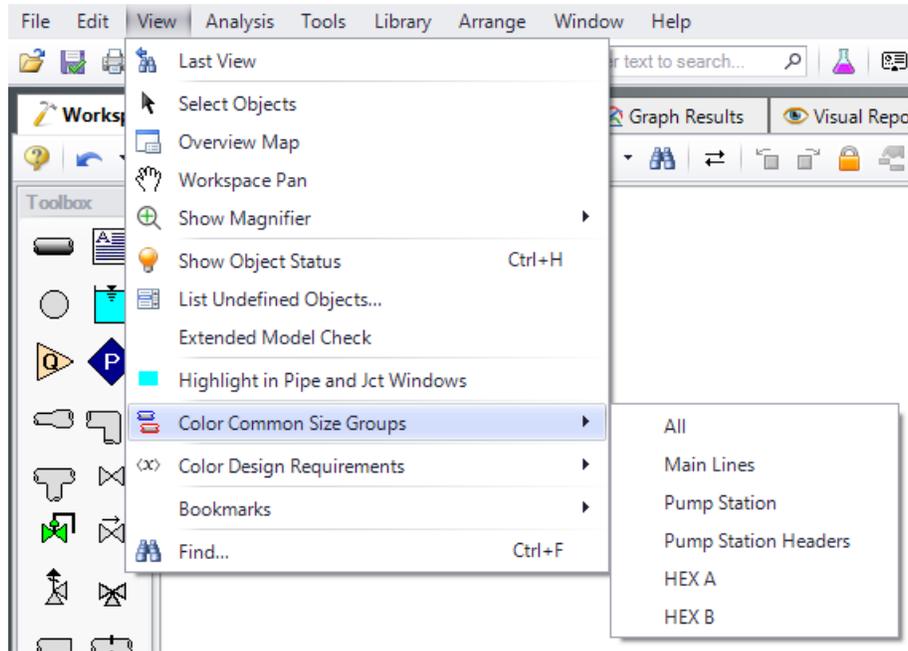


Figure 4: Common Size Groups organized by color

Related Examples

Cooling System - ANS 1176
 Hot Water System - ANS 1217
 Variable Drive - ANS 1231

Junction Loss Factors

During the sizing process, the size of junctions will implicitly change along with the size of the neighboring pipes.

Junctions which use handbook loss factors will automatically adjust their loss value when the pipe size changes. For example, a Bend junction defined as a Smooth Bend uses built-in relationships to determine a loss based on pipe area and r/D . As the pipe size changes, this loss value will change automatically.

However, a built-in loss model that updates for different pipe sizes is not always available. Many junctions allow the specification of a Loss Multiplier Table as seen below.

	Diameter	Multiplier	K
1	4.026	1.2	12
2	6.065001	1	10
3	7.981001	0.60	6
4			
5			
6			
7			

Optional Data
 Diameter Units: inches Edit Table
 Get Nominal Diameter... Calculate Multipliers
 Loss Multipliers

Figure 1: Loss Multiplier Table

With a base loss factor (K-Factor, in Figure 1), ANS can use this table to determine how much to adjust the loss by when changing the associated pipe diameters. This can be done by inputting the Diameter and K-factor columns, specifying one of the rows as a multiplier of one, then using the Calculate Multipliers button below the table.

Improving and Troubleshooting the Sizing Process

Check the Continuous Solution First

For small problems with less than ten independently sized pipes, it should be possible to obtain a Discrete solution in a few minutes or less. Larger problems can take a long time to solve, so it is important to point the sizing process in a good starting direction.

The Discrete problem is generally much more difficult to solve than the Continuous, and most of the Discrete methods depend on an initial Continuous solution. Therefore, it is important to ensure the Continuous solution is good. There are a few guidelines to consider:

- **Check the quality of the Continuous Solution before proceeding to the Discrete Solution** - If the Continuous solution is not ideal, then the methods which use the Continuous solution for a starting point will have a poor Discrete solution as well.
- **Ensure the Discrete Solution is worse than the Continuous Solution** - A Discrete solution always places further limits on the solution, and should therefore have a negative impact on the Objective. If the Discrete solution is better than the Continuous, this means that either the Continuous solution is poor, or that there are numerical issues affecting the solution by some small amount. If the difference is significant, this likely indicates an issue with the sizing process, the solver, or user input.
- **Check for Consistent Continuous Solutions** - Using any of the Continuous methods should give comparable results. If this is not the case, it may indicate that ANS is finding Local Minimums with one or more of the methods.
- **Use the best Continuous Solution to Proceed** - If one of the methods is finding a better Continuous Solution than the others, and the results are reasonable, this result should be used as the basis for the Discrete Solution.

Determine if the solution is a Local Minimum

It is possible for ANS to get stuck in a local minimum and not find the global minimum. If this is happening, there are a few things to try:

- **Change the initial pipe sizes**
- **Use a different Sizing Method** - Pure Discrete Methods are inherently better suited to finding global minimums.
- **Temporarily remove some of the Design Requirements** - This can help identify Design Requirements that are actually driving the sizing process. Having too many Design Requirements can make the system much more difficult to solve with minimal gain.

Use a Different Search Method

The different search methods have different strengths and weaknesses. It is possible that one method will struggle with a problem that others can easily find a solution for.

It is also a good indicator that the best solution has been found if different methods agree.

Use Logical Design Requirements

- **Design Requirements in Closed Areas of a Model** - ANS will attempt to identify Design Requirements that are irrelevant and ignore them during the sizing process. However, depending on ANS to do this is not advised. Design Requirements on closed areas should be removed. For example, it is not meaningful to apply a minimum flow rate requirement on a closed pipe - it is possible that ANS will not find a Feasible solution if such a Design Requirement is left in place.
- **Conflicting Design Requirements** - It is easy to apply requirements that are impossible to meet. For example, a minimum flow rate greater than an in-line Flow Control Valve setpoint, a maximum

pressure lower than the connected Assigned Pressure, or a maximum speed on a pump that has been manually set to a higher speed. With these types of Design Requirements, ANS will (correctly) conclude that no feasible design exists, and the results may not be meaningful.

Change the Initial Pipe Sizes

The sizing process uses the initial pipe sizes defined by the user as a starting point. It is possible to start the sizing process in an area that poses difficulty. Increasing or decreasing the initial pipe sizes can help.

Simplify the Problem

It is good practice to solve a simpler problem first, before proceeding to a more complex problem.

- **Be sure the model solves as expected before using ANS** - Any issues that are present in the base hydraulic model will also be present in the sizing process.
- **Find a Continuous Solution before searching for a Discrete solution.**
- **Use a small number of Design Requirements**, and add them as necessary. For troubleshooting purposes, it can be useful to remove Design Requirements that are necessary in the final design.
- **Decrease the number of variables** - Adding more pipes to Common Size Groups, and limiting the number of Common Size Groups, can make the problem much easier to solve.

Sizing Components Other Than Pipes

Sizing Pumps

While pumps are not directly sized by ANS, they are able to be indirectly sized by including an estimation of their cost. Their actual behavior is a result of the system. The appropriate sizing of a pump requires a [Cost Library](#) and understanding of standard [Pump Sizing](#).

At the most basic, pump sizing is simply selecting a pump that is capable of meeting pressure and flow requirements. Too often this involves selecting a pumps that is larger than required, under the assumption that a bigger pump will provide for operational margin. Unfortunately, over-sizing a pump has negative impacts on [reliability](#) and [efficiency](#), both of which incur higher cost. Over-sizing could also require the installation of ultimately unnecessary valves or restrictions to bring the flow back down to where it should be.

The ability to appropriately size pumps is built into AFT Fathom. By using the [Pump Sizing](#) option, AFT Fathom will indicate the amount of head required to deliver the specified flow. It is up to the user to find a pump that operates at its BEP at the calculated flow and head.

The best size of pump to use when the pipe sizes are allowed to change as well is harder to handle, and requires iteration. See [Sizing Pipes and Pumps Concurrently](#).

Note: Certain pump configurations require special attention in ANS. See [Special Concerns For Sizing Pumps](#).

Sizing Control Valves

Control Valve sizing is similar - AFT Fathom already has all of the tools needed to size a Control Valve for a specific set of pipes. AFT Fathom will report the Cv or K Factor necessary for a Control Valve to attain a given setpoint. From this value, an appropriate valve could be selected for minimizing cost. It would be typical to select the smallest Control Valve possible while still remaining in a good control range - typically 20-80% open.

Like pumps, selecting the best Control Valve while pipe sizes are changing is more complex. See [Sizing Pipes and Control Valves Concurrently](#).

Sizing Other Junctions

Any other junction can be included in the sizing process as long as cost data exists.

For example, a Bend can have costs associated with it via a [Table of Costs](#). From this table, ANS can determine the cost for a given diameter. ANS will assume that the Bend is of a diameter equal to the upstream pipe inside diameter, and include the corresponding cost in the Objective. If certain components are very expensive compared to piping, this could affect the selection of pipe sizes.

Sizing Ducts

Duct sizing has special considerations discussed in [Sizing Duct Systems](#).

Related Topics

Special Concerns For Sizing Pumps.....	667
Sizing Pipes and Pumps Concurrently.....	669
Sizing Pipes and Control Valves Concurrently.....	673
Sizing Pipes, Pumps, and Control Valves Concurrently.....	676
Sizing Duct Systems.....	678

Special Concerns For Sizing Pumps

There are several special concerns with sizing pumps. For a general overview of the process of sizing pumps see: [Sizing Pipes and Pumps Concurrently](#).

End of Curve Flow Rate

Centrifugal pumps typically have an end of curve, or runout, flow rate. They cannot operate reliably past this point. This can be entered in [Pump Configuration](#), but this will not actually prevent the model from running the pump with higher flows. Instead, the curve will be extrapolated past this point and a warning will be generated.

If it is important to prevent the sizing results from considering flows in this region as valid, a Design Requirement for maximum flow rate equal to the end of curve value should be placed on the pump.

Partial Service Duration

Without other input, ANS will assume that any pump included in the sizing operates constantly.

If this is not the case, then either an appropriate Service Duration or Multiplier should be defined in Assign Cost Libraries.

Variable Speed Pumps with Partial Control Applicability

ANS can size pumps with a user defined fixed speed other than 100%, or a controlled pump that modifies speed to meet a flow or pressure setpoint. However, pressure-type variable speed controlled pumps allow the user to optionally enforce Control only when the actual pressure is above/below a setpoint. This causes an issue with sizing.

If ANS allowed the pump to change from a fixed 100% speed pump (non-controlling pressure) to a variable speed pump (when the setpoint is violated), it would be changing the characteristics of the system. This has a direct impact on the sizing process, so it cannot be allowed - ANS will **always** model this type of pump as fixed 100% speed.

Either an Always Controlling pump must be specified, or Design Requirements on pressure must be added to the appropriate pipes.

Unrealistic Sizing Results From Dependent Design Cases

If there are two design cases that have different pump flow rate requirements, the ideal pump curve may be unrealistic.

For example, one operating case may require 500 gal/min, and another may require 750 gal/min. It is desired to select the ideal pump. After running ANS, it is determined that the pump should generate 50 feet of head in the 500 gal/min case and 70 feet in the 750 gal/min case. To meet this requirement, we need a pump curve that increases from 50 feet of head to 70 feet with a large increase in flow. This is an unrealistic demand.

To prevent this, a head rise Design Requirement can be added to maintain the high flow head at 70-90% of the low flow head. This will effectively force the sizing to consider only realistically shaped pump curves.

Related Topics

Sizing Components Other Than Pipes.....	666
Sizing Pipes and Pumps Concurrently.....	669
Sizing Pipes and Control Valves Concurrently.....	673
Sizing Pipes, Pumps, and Control Valves Concurrently.....	676
Sizing Duct Systems.....	678

Related Examples

Cooling System - ANS.....	1176
---------------------------	------

Sizing Pipes and Pumps Concurrently

The cost of a pumped piping system clearly depends on both the cost of the pipes, and the pump. It is tempting to select pumps and pipes in stages, but this is unlikely to produce the lowest cost solution. Selecting a smaller pump may not appear to be viable, but changing the pipe sizes may make it viable. Like sizing pipes alone, overall costs can often be decreased by increasing the cost of one or a few components. Attempting to find the lowest cost option manually quickly becomes impractical even with a small number of variables.

Optimal Pumping System Operating Point (OPSOP)

A pump's BEP is considered the ideal place to run a pump as the efficiency is highest, and thus cost is lowest. Similarly, any piping system will have a lowest cost design.

Whether a pump is running at BEP, however, depends on the piping system, and the lowest cost piping system depends on the pumps it contains. What may be the lowest cost piping system *for a given pump*, may not be the lowest *possible* cost option. It is possible that a cheaper pump could be selected with the correct piping adjustments while still maintaining all Design Requirements.

The point at which no adjustments to pump or system will result in lower cost is known as the **Optimal Pumping System Operating Point (OPSOP)**. One way to determine this point is to exhaustively size the system for different potential sizes of pumps. This is inefficient, but can be used to construct a visual representation of the OPSOP as shown below.

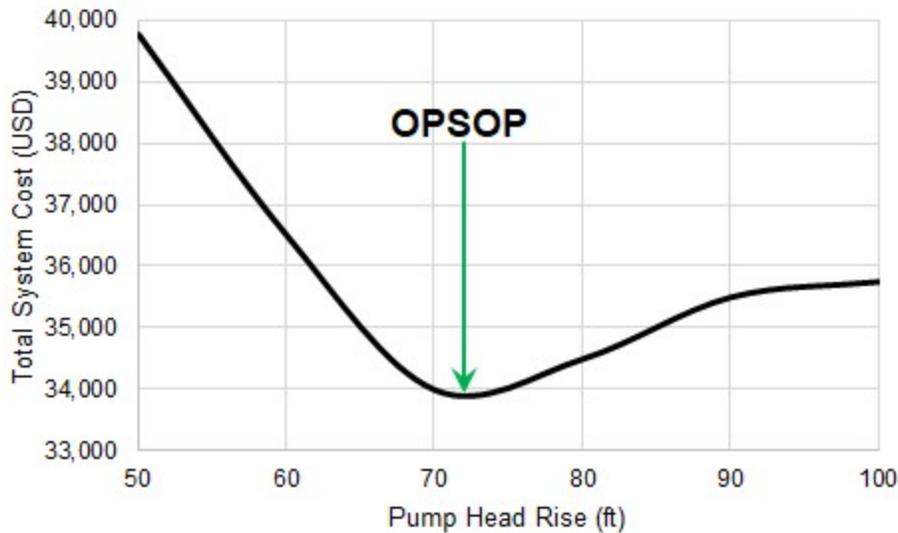


Figure 1: The OPSOP of a given system. Note that every point is an ideal sizing solution for the given pump size

If the sizing process is approached in stages, rather than concurrently, it is easy to get a result that says, for example, a 60 foot head rise pump is required. The **best case** system for the 60 foot head rise pump has a minimum cost of about \$36,500. However, allowing the pump to change as well can result in a system more than 7% cheaper.

Process Outline

Find the Approximate Pump Size

1. Model the pump with the [Pump Sizing](#) option, fixing the flow to the desired amount. AFT Fathom will determine the required head rise.
 - Include an estimate of Nominal Efficiency.
 - Optionally, include Nominal NPSHR.
2. Add the pump to an [Engineering Library](#).
 - Attach a [Cost Library](#) to the Engineering Library if one does not already exist.
 - Create a Power vs. Cost [Scale Table](#) for the pump with estimated cost values.
3. Select the appropriate options in the [Sizing Window](#).
 - The cost for the pump will only be included in the [Objective](#) if it is specified to be included in [Size/Cost Assignments](#).
 - If there are several pumps that will ultimately be of the same type, they should be placed into a Maximum Cost Group.
4. Run ANS.

- Investigate the resulting pump size.
- Update the Nominal Efficiency and NPSHR based on the result. Also update the cost information if necessary. If the pump accounts for a large portion of the overall cost, accurate values here are more important.
- Repeat the process as necessary, until reasonable estimates for cost at the resulting operating point are obtained.

Compare Candidate Pumps

1. Locate accurate cost information and [pump curves](#) for the candidates.
 - Incorporate these values into Engineering and Cost Libraries.
2. Compare the sizing results for each candidate pump.

Estimating Pump Costs

Without knowing specifically what type or size of pump is going to be included in the system, the pump behavior must be approximated. The behavior is approximated with the [Pump Sizing](#) option, indicating a design flow. We also need to approximate cost. Because energy cost is often the largest component of a pump's overall cost, and this cost is directly related to the pump's power, it makes sense to correlate other pump costs to power as well.

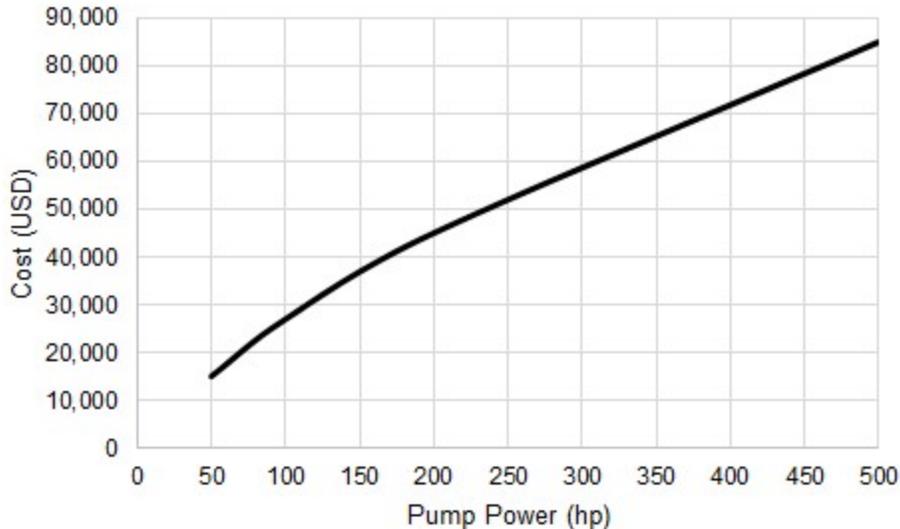


Figure 2 - Total Initial Costs for a series of pumps

Keep in mind that such a comparison is only valid for the selected range of pumps. Different designs of pumps, or pumps from different manufacturers may follow different relationships. One manufacturer may offer cheaper low-power pumps, while their competitor offers cheaper high-power pumps. This cost information is represented in ANS as a Power vs Cost [Scale Table](#). This requires the creation of a [Cost Library](#) including the pump as a [Custom Component](#).

Because we do not know what specific pump is going to be used, we also do not know exactly how much it will cost to run or maintain it.

As part of our sizing process, we must assume that the pump runs at some **Nominal Efficiency**. AFT Fathom will calculate the fluid power required at any given flow and head rise. However, a pump or motor is not perfectly efficient and will therefore require more power than the fluid power. This affects the pump selection, and therefore cost, so it is important to include an estimate on efficiency.

Note: It may seem that including a nominal efficiency is not important, as selecting a certain pump size will minimize the initial pump cost regardless of this value. However, this does not consider the cost of power over a long period, which is often a driving factor in sizing. Also, it is important to get the relative cost of the pump correct as this can affect the best sizing. The nominal efficiency will require more power and hence a more expensive pump - as this is included in the [Objective](#), it is important to account for.

If maintenance costs are expected to be significant, it is important to include an estimate of these costs. Again, this could be accounted for with a Power vs Cost Scale Table.

The Approximate Sizing Solution

When the initial model is defined, the sizing configuration complete, and the pumps appropriately defined with approximate cost information, the automated sizing can take place.

ANS will size the pipes of the system to minimize cost while obeying Design Requirements. It will now include a power-based cost for the pump. Without cost information on the pump, ANS would tend to select small pipe sizes to reduce cost without any negative consequence. However, smaller pipe sizes have higher frictional loss, which requires greater head rise at the pump, and hence higher cost. By including pump cost information, the balance between pump costs and piping costs can be found.

However, this is still an **estimate** because we are not modeling a real pump. Fixing the flow and nominal efficiency are good design tools but it is important to represent a real pump curve not only to account for real effects, but also to ensure that the desired pump can actually be acquired.

If the pump costs are a significant portion of the overall cost, the estimates made for cost and nominal efficiency should be refined and the system re-sized before proceeding.

Sizing with a Real Pump

Now that there is a good idea of the size and cost of pump we are looking for, we can select a real pump we may want to use.

It will rarely be clear which pump is the best choice. First, the exact operating point determined from the approximate solution is likely unavailable in a commercially available pump. Second, different pumps have different costs - even if a pump with the exactly correct specifications is available, it may be much more expensive than a pump that is slightly mismatched. This cost difference could mean the cheaper pump is better overall, despite not appearing to be the ideal choice.

The best course is to select several candidate pumps, and size the system independently with each candidate pump. It is important that accurate [pump curves](#) and cost information be included for each can-

didate, as this no longer meant to be an estimation. After testing each candidate pump, the system with the lowest overall cost can be selected.

If the sizing results differ significantly in this stage this could indicate that the pump is poorly matched, that there was an error in the process, or that the system is dominated by pump costs and therefore very sensitive to the estimations.

Related Topics

Sizing Components Other Than Pipes.....	666
Special Concerns For Sizing Pumps.....	667
Sizing Pipes and Control Valves Concurrently.....	673
Sizing Pipes, Pumps, and Control Valves Concurrently.....	676
Sizing Duct Systems.....	678

Sizing Pipes and Control Valves Concurrently

Process Outline

Find the Approximate Control Valve Size

1. Model the Control Valve with the appropriate settings. AFT Fathom will determine the required loss.
2. Add the Control Valve to an [Engineering Library](#).
 - Attach a [Cost Library](#) to the Engineering Library if one does not already exist.
 - Create a C_v vs. Cost [Scale Table](#) for the control valve with estimated cost values.
3. Select the appropriate options in the [Sizing Window](#).
 - The cost for the control valve will only be included in the [Objective](#) if it is specified to be included in [Size/Cost Assignments](#).
 - If there are several control valves that will ultimately be of the same type, they should be placed into a Maximum Cost Group.
4. Run ANS.
 - Investigate the resulting control valve size.
 - Update the cost information if necessary. If the control valve accounts for a large portion of the overall cost, accurate values here are more important.
 - Repeat the process as necessary, until reasonable estimates for cost at the resulting operating point are obtained.

Compare Candidate Control Valves

1. Locate accurate loss and cost information for the candidates.
 - Incorporate these values into Engineering and Cost Libraries.
2. Compare the sizing results for each candidate control valve.

Reference Pressure Concerns

It is impossible to generate a unique solution from a system (or part of a system) entirely bounded by fixed flows. For an in-depth discussion of this issue and common areas where it may occur see [Role of Pressure Junctions - Detailed Discussion](#).

Particularly common when sizing a system is the tendency to place a Flow Control Valve immediately downstream of a Pump modeled as a fixed flow. This situation has no unique solution, as discussed in the [Sizing Pumps with Flow Control Valves](#) section of the previous topic.

The typical solution to this issue is to temporarily change the Flow Control Valves to Pressure Drop Control Valves where necessary, until a pump curve is selected and specified. Effectively, this is the equivalent of specifying that our Flow Control Valve has a pressure drop requirement equal to the Pressure Drop Control Valve setpoint.

Control Valves and Design Requirements

ANS will *always* attempt to minimize or maximize the value of the Objective. **ANS does not know if the resulting design is realistic and this has no impact on the sizing.** It can present results that would be physically impossible to attain in the field.

AFT Fathom is no different in this regard - it is possible to specify a system that would require physically unrealistic behavior. For example, the liquid pressures are allowed to drop below vapor pressure. When possible, there will be Warnings if this type of behavior is detected, but the solver does not make any attempt to *prevent* this situation. To avoid these unrealistic configurations, the user must inspect the output and determine the changes required on the system to avoid the situation. In other words, the user must adjust the design of the system to meet desired or realistic behavior - a Design Requirement.

As an analysis tool, AFT Fathom does not allow the user to directly impose these requirements. ANS, on the other hand, is a design tool and actively selects inputs to meet certain specifications. It would not be unusual to specify a minimum pressure Design Requirement to prevent the vapor pressure issue.

How does this relate to Control Valves? Control Valves in AFT Fathom will, by default, [force control](#). This means that the system will be able to behave unrealistically in order to meet the setpoint specified, which can be useful in a design stage. **As it would not be useful for a Control Valve to be unable to achieve its setpoint during the sizing process, Always Control (Never Fail) is automatically enabled for sizing.**

Consider a simple case of a Flow Control Valve regulating flow between two reservoirs. It is desired to size this system for lowest pipe weight.

Because of Always Control (Never Fail), the Flow Control Valve is essentially able to act like a pump - providing "free" pressure to the system in order to meet the setpoint. As the cost of this "pump" is not considered, the sizing process is trivial - we can simply select the smallest pipe size available, as our "pump" will simply make up for the increase in pressure loss.

Clearly, it is not realistic for a Control Valve to add pressure to a system. So why don't we allow our Flow Control Valve to not meet its setpoint, as it would in reality? If the Control Valve is allowed to do so, we are essentially stating that the flow it is attempting to maintain is not actually a requirement. If this is the case, then again we will find that the "ideal" sizing solution is to select the smallest pipe size available. The Flow Control Valve will fully open in an attempt to allow more flow, but in the end the flow will be below the setpoint. However, our pipe weight will certainly be minimized.

What is the solution? **We need to add a Design Requirement.** It may seem intuitive to add a minimum flow requirement to our system as we are attempting to control flow. However, this will not solve the problem because the Control Valve will still simply add pressure to meet the setpoint. Furthermore, adding Design Requirements that [conflict](#) with the model itself can cause problems.

Instead, a Design Requirement should be added related to the *position* of the Control Valve. The position can be represented either directly as an Open Percentage if that data is available, or as a form of loss. It is typical that a Control Valve is designed to operate within a certain range of positions - often 20-80% open. It would not be useful to have a Control Valve designed to operate at 100% open, as a slight variation in flow could cause it to not meet its setpoint and open fully.

By adding a Design Requirement on minimum dP/dH or maximum C_V /Open Percentage we prevent unrealistic behavior and return a meaningful sizing result.

Note: It is **always** required to add this Design Requirement for Control Valves in the sizing model - even if the junction is not marked to be Sized in Size/Cost Assignments. Including a Control Valve in Sizing adds its cost to the Objective which can influence the result, which is not always a concern. However, the behavior described above is always true and hence always needs a Design Requirement.

Optimal Control System Operating Point

Just like [sizing pipes and pumps concurrently](#), there is an optimal size of a control valve that minimizes total system cost. Approaching the problem in "stages" rather than concurrently can easily miss the lowest overall cost solution.

Like the pump process, we do not know specifically what Control Valve is going to be used in the early stages of the process so we need to estimate the cost.

For Control Valves, the logical choice is to estimate with a Cost vs C_V relationship. This information will be represented for the particular component in a Scale Table.

Note: When defining a Cost vs C_V Scale Table, keep in mind that this table is relating the **current** C_V to the cost of the device. For example, if the Control Valve has a Design Requirement to be 80% open, and ANS calculates a corresponding C_V of 120, the cost used from the Scale Table will be that for a C_V of 120, **not** the full-open value of the control valve. This is important to keep in mind when creating the scale table, as manufacturer cost information will often be in the form of full-open C_V vs cost.

The Approximate Sizing Solution

When the initial model is defined, the sizing configuration complete, and the control valves appropriately defined with approximate cost information, the automated sizing can take place.

ANS will size the pipes of the system to minimize cost while obeying Design Requirements - including the Design Requirement on position discussed above. However, this is still an **estimate** because we are not modeling a real control valve.

If the control valve costs are a significant portion of the overall cost, the estimates made for cost should be refined and the system re-sized before proceeding.

Sizing with a Real Control Valve

Now that there is a good idea of the size and cost of control valve we are looking for, we can select a real control valve we may want to use.

It will rarely be clear which control valve is the best choice. First, the exact operating point determined from the approximate solution is likely unavailable commercially. Second, different valves have different costs - even if a valve with the exactly correct specifications is available, it may be much more expensive than one that is slightly mismatched. This cost difference could mean the cheaper valve is better overall, despite not appearing to be the ideal choice.

The best course is to select several candidate valves, and size the system independently with each candidate. It is important that accurate loss and cost information be included for each candidate, as this no longer meant to be an estimation. After testing each candidate, the system with the lowest overall cost can be selected.

If the sizing results differ significantly in this stage this could indicate that the control valve is poorly matched, that there was an error in the process, or that the system is dominated by control valve costs and therefore very sensitive to the estimations.

Related Topics

Sizing Components Other Than Pipes.....	666
Special Concerns For Sizing Pumps.....	667
Sizing Pipes and Pumps Concurrently.....	669
Sizing Pipes, Pumps, and Control Valves Concurrently.....	676
Sizing Duct Systems.....	678

Related Examples

Control Valve - ANS	1149
Hot Water System - ANS	1217

Sizing Pipes, Pumps, and Control Valves Concurrently

Many systems contain both pumps and control valves, which makes the sizing process somewhat more complex.

Like sizing [pipes and pumps](#) or [pipes and control valves](#) concurrently, there are adjustments and estimations that must be made when sizing all three in parallel. The overall goal and considerations that must be taken into account are described in detail in those topics and will not be repeated here. Only the outline of steps will be presented for brevity.

Process Outline

Find the Approximate Pump and Control Valve sizes

1. Model the pump(s) with the [Pump Sizing](#) option, fixing the flow to the desired amount.
2. Model the Control Valve(s) with the appropriate settings.
 - If the control valve(s) are Flow Controls downstream of the pump(s) to be sized, temporarily set them to be Pressure Drop Control Valve with a setpoint equal to the minimum required pressure drop.
 - Any non-Pressure Drop control valves will require a Design Requirement of minimum dP/dH or maximum Cv/Open Percentage.
3. Add the pump(s) and control valve(s) to one or several [Engineering Libraries](#).
 - Attach a [Cost Library](#) to the Engineering Library if one does not already exist.
 - Create a Power vs. Cost [Scale Table](#) for the pump with estimated cost values.
 - Create a Cv vs. Cost [Scale Table](#) for the control valve with estimated cost values.
4. Select the appropriate options in the [Sizing Window](#).
 - The cost for the pump(s) and control valve(s) will only be included in the [Objective](#) if it is specified to be included in [Size/Cost Assignments](#).
 - If there are several pumps or control valves that will ultimately be of the same type, they should be placed into Maximum Cost Groups.
5. Run ANS.
 - Investigate the resulting pump size.
 - Update the Nominal Efficiency and NPSHR based on the result. Also update the cost information if necessary. If the pump accounts for a large portion of the overall cost, accurate values here are more important.
 - Repeat the process as necessary, until reasonable estimates for cost at the resulting operating point are obtained.

Compare Candidate Pumps

1. Locate accurate cost information and [pump curves](#) for the candidates.
 - Incorporate these values into Engineering and Cost Libraries.
2. Compare the sizing results for each candidate pump.

Select the two or three best candidate pumps and compare candidate Control Valves.

1. If necessary, change Pressure Drop Control Valves back to Flow Control Valves.
 - This will require a Design Requirement of minimum dP/dH or maximum Cv/Open Percentage.
2. Locate accurate loss and cost information for the candidate control valves.
 - Incorporate these values into Engineering and Cost Libraries.
3. Compare the sizing results for each candidate control valve, under each candidate pump.

Related Topics

Sizing Components Other Than Pipes.....	666
Special Concerns For Sizing Pumps.....	667
Sizing Pipes and Pumps Concurrently.....	669

Sizing Pipes and Control Valves Concurrently.....	673
Sizing Duct Systems.....	678

Related Examples

Cooling System - ANS	1176
----------------------------	------

Sizing Duct Systems

A [Pipe Material Library](#) of all possible rectangular duct sizes is needed, as is the case with Cylindrical pipe.

If there are no geometric restrictions, the [Candidate Set](#) should use all of the duct sizes. Certain areas of the system may have maximum width or height restrictions. In these areas, a different Candidate Set should be created from the same Pipe Material Library.

There are two variables for a rectangular duct - height and width, whereas a normal pipe only has one - diameter. ANS treats these variables as independent when solving for the ideal size. This means that duct Candidate Sets must have corresponding widths and heights for every duct dimension.

For example, consider 3 duct sizes in a Candidate Set: 8x8, 10x10, and 12x12. Because width and height are considered independent by ANS, it is possible that 8 would be selected for width and 10 for height. However, 8x10 is not in the Candidate Set. ANS will warn that the Candidate Set is incomplete when this is detected.

Related Topics

Sizing Components Other Than Pipes.....	666
Special Concerns For Sizing Pumps.....	667
Sizing Pipes and Pumps Concurrently.....	669
Sizing Pipes and Control Valves Concurrently.....	673
Sizing Pipes, Pumps, and Control Valves Concurrently.....	676

Search Methods

ANS includes several different Search Methods for finding the best sizing solution. Some methods are very similar, while others are entirely unique, and each is particularly suited to certain kinds of mathematical problems.

It is impossible to state that one method is always better or worse than another. The quality and speed of each method depends on the specific system being solved and the sizing input. A method that excels for one model may not be the best choice for another model.

Note: All of the methods share a common goal of minimizing (or maximizing) an Objective. The analytical approach driving the sizing (the hydraulic solver) does not vary with the selected method. This means that the best sizing solution across all of the methods is the best overall solution. **It is good practice to test several of these methods for any model.**

- [Modified Method of Feasible Directions](#) - Reliable and applicable to most problems.
- [Sequential Quadratic Programming](#) - Frequently the most efficient, but more sensitive to system definition.
- [Sequential Unconstrained](#) - Sensitive to initial sizes and control parameters, but more capable of handling large numbers of uniquely sized pipes.
- [Genetic Algorithm](#) - Cross-breeds a random "population" of designs, mimicking biological evolution. Less likely to become trapped in a local minimum, but very slow compared to other methods.

Modified Method of Feasible Directions

A [Continuous Search Method](#) available in the [Sizing Method panel](#).

The Modified Method of Feasible Directions (MMFD) finds the best case design by determining a *search direction* based on the [Objective](#) function and the [active Design Requirements](#). When there are no active Design Requirements, the search direction will simply be in a direction influenced by the current [gradient](#) and previous search directions.

When there is an active Design Requirement, following the direction of greatest change may make the design [Infeasible](#). Instead, the search direction is modified to keep the solution in the feasible region. If a Design Requirement is Violated, the search direction will attempt to return the solution to the feasible region as quickly as possible.

This is the most robust method and can handle a wide variety of problems.

Note: [Branch and Bound](#) rapidly becomes inefficient with an increase in independent variables. For [discrete](#) optimization with more than 10 independently sized pipes, the [Sequential Unconstrained](#) or [Genetic Algorithm](#) methods should be considered.

Sizing Control Parameters

- **Finite Difference Gradient Calculation** - The most likely parameters to be changed, gradient calculation directly influences the search direction. See a detailed discussion in [Estimating the Gradient](#).
 - **Method** - [Forward or Central](#), directly affects the quality of the gradient at the cost of runtime. Central gradients are more accurate but require about twice as many hydraulic solutions.
 - **Step Size** - How much to perturb the pipe sizes to determine the gradient. If the Objective was known exactly, it would make sense to make these values very small. However, as Fathom itself is an [iterative solver](#), the Objective is only known to within a certain tolerance. Making the Finite Difference Step Size too small actually introduces uncertainty into the gradient and can therefore generate poor search directions making the process take longer, if a solution can be found at all. A Step Size that is too large has a similar effect, so it is important to select an appropriate value between the two extremes. The best value to use depends on the system in question - when in doubt, use the default.
- **Print Text Debug File** - Generate a tab-separated file reporting on each call to the hydraulic solver. Actual pipe sizes are presented along with current Objective value and Design Requirement status.

Caution: The Advanced Control Parameters below should not be changed under normal circumstances without the direction of AFT Support.

- **Sizing Level Convergence**
 - **Maximum Iterations** - This value is the number of *sizing iterations*, not the number of hydraulic solutions allowed. Generally, the hydraulic solution is by far the slowest part of the process, and as the number of uniquely sized pipes increases, so does the number of required hydraulic solutions. A problem with 100 independently sized pipes would require, at minimum, 100 hydraulic solutions just for 1 sizing iteration. That said, it is important to have an adequate number of sizing iterations regardless of the number of variables - if the process is stopped prematurely, the ideal sizes have not been found.
 - **Consecutive Iterations for Convergence** - The sizing process will be repeated at least this number of times while satisfying the convergence criteria. It is helpful to allow additional iterations because the sizing process can occasionally make little change in one iteration and then significant change in the next.
 - **Relative Tolerance** - Changes smaller than this relative amount (default of 0.1%) are considered small enough to be converged. Note that the number of iterations requiring a change in the value of the Objective smaller than this amount can be modified.
 - **Absolute Tolerance** - The value entered here is multiplied by the initial Objective value. The resulting value is considered as an absolute tolerance. By default, the Objective is scaled to 1,000 meaning that the default value of 0.0001 results in an absolute tolerance of $1,000 \times 0.0001 = 0.1$. Objective values smaller than 0.1 will use the absolute criterion instead of the relative.
- **Sequential Unconstrained Parameters**
 - **Sequential Unconstrained Parameters** - See [Sequential Unconstrained](#).
- **Design Requirement Tolerances**
 - **Violated** - A very small positive value, which allows Design Requirements to be violated. It is important to allow this small violation for numerical reasons, as attempting to force values to zero is often difficult computationally.
 - **Active** - This small negative number determines how close to the Design Requirement the Objective can be before the Design Requirement influences the search direction. This value is updated automatically during the sizing process.
- **Scaling Parameters**
 - **Desired Objective Scaled Value** - In some cases it is helpful to scale the value of the Objective. Very large or very small unscaled Objective values can present numerical issues, and scaling can help with this.
 - **Scale Factor for Design Requirements**
 - **Common** - Design Requirements are always scaled (dividing by the user defined quantity) for numerical reasons. This factor allows additional scaling.
 - **Violated Tolerance** - This can be used to reduce the value of Violated Design Requirement Tolerance. This can be useful if the small violation is unacceptable for a feasible design.
- **Print Optimizer Debug File** - Generate a debug file detailing the normalized and generalized optimization process.

Related Topics

Search Methods.....	678
Sequential Quadratic Programming.....	681
Sequential Unconstrained.....	683
Genetic Algorithm.....	686

Related Examples

Beginner - Three Reservoir Problem - ANS	1134
Control Valve - ANS	1149
Cooling System - ANS	1176
Hot Water System - ANS	1217
Housing Project - ANS	1246
Variable Drive - ANS	1231

Sequential Quadratic Programming

A [Continuous Search Method](#) available in the [Sizing Method panel](#).

Sequential Quadratic Programming (SQP) finds the best case design by solving a series of quadratic approximations to the Objective Function. Each of these "Sub-Problems" is solved with the [Modified Method of Feasible Directions](#). This method is often the best and most efficient if the problem is numerically well defined. Sizing with this method is always worth checking.

Note: [Branch and Bound](#) rapidly becomes inefficient with an increase in independent variables. For [discrete](#) optimization with more than 10 independently sized pipes, the [Sequential Unconstrained](#) or [Genetic Algorithm](#) methods should be considered.

Sizing Control Parameters

- **Finite Difference Gradient Calculation** - The most likely parameters to be changed, gradient calculation directly influences the search direction. See a detailed discussion in [Estimating the Gradient](#).
 - **Method** - [Forward or Central](#), directly affects the quality of the gradient at the cost of runtime. Central gradients are more accurate but require about twice as many hydraulic solutions.
 - **Step Size** - How much to perturb the pipe sizes to determine the gradient. If the Objective was known exactly, it would make sense to make these values very small. However, as Fathom itself is an [iterative solver](#), the Objective is only known to within a certain tolerance. Making the Finite Difference Step Size too small actually introduces uncertainty into the gradient and can therefore generate poor search directions making the process take longer, if a solution can be found at all. A Step Size that is too large has a similar effect, so it

is important to select an appropriate value between the two extremes. The best value to use depends on the system in question - when in doubt, use the default.

- **Print Text Debug File** - Generate a tab-separated file reporting on each call to the hydraulic solver. Actual pipe sizes are presented along with current Objective value and Design Requirement status.

Caution: The Advanced Control Parameters below should not be changed under normal circumstances without the direction of AFT Support.

- **Sizing Level Convergence**
 - **Maximum Iterations** - This value is the number of *sizing iterations*, not the number of hydraulic solutions allowed. Generally, the hydraulic solution is by far the slowest part of the process, and as the number of uniquely sized pipes increases, so does the number of required hydraulic solutions. A problem with 100 independently sized pipes would require, at minimum, 100 hydraulic solutions just for 1 sizing iteration. That said, it is important to have an adequate number of sizing iterations regardless of the number of variables - if the process is stopped prematurely, the ideal sizes have not been found.
 - **Consecutive Iterations for Convergence** - The sizing process will be repeated at least this number of times while satisfying the convergence criteria. It is helpful to allow additional iterations because the sizing process can occasionally make little change in one iteration and then significant change in the next.
 - **Relative Tolerance** - Changes smaller than this relative amount (default of 0.1%) are considered small enough to be converged. Note that the number of iterations requiring a change in the value of the Objective smaller than this amount can be modified.
 - **Absolute Tolerance** - The value entered here is multiplied by the initial Objective value. The resulting value is considered as an absolute tolerance. By default, the Objective is scaled to 1,000 meaning that the default value of 0.0001 results in an absolute tolerance of $1,000 * 0.0001 = 0.1$. Objective values smaller than 0.1 will use the absolute criterion instead of the relative.
- **Sub-Problem Iterations**
 - **Maximum** - This is the maximum number of approximate problem cycles.
 - **Consecutive for Convergence** - The approximations will be repeated at least this number of times while satisfying the Sizing Level Convergence tolerance criteria.
- **Sequential Unconstrained Parameters**
 - **Maximum Number of Independent Pipe Sizes for Branch and Bound** - Discrete solutions with this method require the use of Branch and Bound. If the system has more independently sized pipes than this value, the Sequential Unconstrained will automatically be used.
 - **Sequential Unconstrained Parameters** - See [Sequential Unconstrained](#).
- **Design Requirement Tolerances**
 - **Violated** - A very small positive value, which allows Design Requirements to be violated. It is important to allow this small violation for numerical reasons, as attempting to force values to zero is often difficult computationally.
 - **Active** - This small negative number determines how close to the Design Requirement the

Objective can be before the Design Requirement influences the search direction. This value is updated automatically during the sizing process.

- **Scaling Parameters**
 - **Desired Objective Scaled Value** - In some cases it is helpful to scale the value of the Objective. Very large or very small unscaled Objective values can present numerical issues, and scaling can help with this.
 - **Scale Factor for Design Requirements**
 - **Common** - Design Requirements are always scaled (dividing by the user defined quantity) for numerical reasons. This factor allows additional scaling.
 - **Violated Tolerance** - This can be used to reduce the value of Violated Design Requirement Tolerance. This can be useful if the small violation is unacceptable for a feasible design.
- **Print Optimizer Debug File** - Generate a debug file detailing the normalized and generalized optimization process.

Related Topics

Search Methods.....	678
Modified Method of Feasible Directions.....	679
Sequential Unconstrained.....	683
Genetic Algorithm.....	686

Sequential Unconstrained

A [Continuous Search Method](#) available in the [Sizing Method panel](#). This method is unique in that it has an internal method to generate [Discrete](#) solutions as well.

Sequential Unconstrained (SU) finds the best case design by solving a series of [unconstrained](#) problems. This simplification is appealing because it is almost always easier to solve an unconstrained problem, making it possible to efficiently solve very large and complex problems. In order to account for the [Design Requirements](#), the [Objective](#) function is artificially increased in [Infeasible](#) regions. This has the effect of driving the unconstrained solution into the feasible region.

Artificially increasing the Objective a large amount on the first cycle creates a numerically ill-conditioned problem which is difficult to solve. Instead, only a small penalty is added initially. This means it is likely the solution on the first cycle will be infeasible. The next cycle increases the penalty factor and uses the results of the previous cycle as a starting point. In this way, the optimum point is moved toward the feasible region in small, numerically stable, steps. Once the result of an unconstrained cycle ends in the feasible region, the process is stopped.

A Hybrid Method

The Modern Penalty Method does not use [Branch and Bound](#) for the determination of Discrete sizes. Instead, it can force the continuous solution into discrete "slots" by creating an artificial set of constraints after the continuous solution is complete. This prevents the need for searching an exceptionally large

"tree" that would result with Branch and Bound. However, the method is generally only effective at finding discrete sizes +/-1 size away from the continuous solution.

Sizing Control Parameters

- **Finite Difference Gradient Calculation** - The most likely parameters to be changed, gradient calculation directly influences the search direction. See a detailed discussion in [Estimating the Gradient](#).
 - **Method** - [Forward or Central](#), directly affects the quality of the gradient at the cost of runtime. Central gradients are more accurate but require about twice as many hydraulic solutions.
 - **Step Size** - How much to perturb the pipe sizes to determine the gradient. If the Objective was known exactly, it would make sense to make these values very small. However, as Fathom itself is an [iterative solver](#), the Objective is only known to within a certain tolerance. Making the Finite Difference Step Size too small actually introduces uncertainty into the gradient and can therefore generate poor search directions making the process take longer, if a solution can be found at all. A Step Size that is too large has a similar effect, so it is important to select an appropriate value between the two extremes. The best value to use depends on the system in question - when in doubt, use the default.
- **Print Text Debug File** - Generate a tab-separated file reporting on each call to the hydraulic solver. Actual pipe sizes are presented along with current Objective value and Design Requirement status.

Caution: The Advanced Control Parameters below should not be changed under normal circumstances without the direction of AFT Support.

- **Sizing Level Convergence**
 - **Maximum Iterations** - This value is the number of *sizing iterations*, not the number of hydraulic solutions allowed. Generally, the hydraulic solution is by far the slowest part of the process, and as the number of uniquely sized pipes increases, so does the number of required hydraulic solutions. A problem with 100 independently sized pipes would require, at minimum, 100 hydraulic solutions just for 1 sizing iteration. That said, it is important to have an adequate number of sizing iterations regardless of the number of variables - if the process is stopped prematurely, the ideal sizes have not been found.
 - **Consecutive Iterations for Convergence** - The sizing process will be repeated at least this number of times while satisfying the convergence criteria. It is helpful to allow additional iterations because the sizing process can occasionally make little change in one iteration and then significant change in the next.
 - **Relative Tolerance** - Changes smaller than this relative amount (default of 0.1%) are considered small enough to be converged. Note that the number of iterations requiring a change in the value of the Objective smaller than this amount can be modified.
 - **Absolute Tolerance** - The value entered here is multiplied by the initial Objective value. The resulting value is considered as an absolute tolerance. By default, the Objective is scaled to 1,000 meaning that the default value of 0.0001 results in an absolute tolerance of

$1,000 * 0.0001 = 0.1$. Objective values smaller than 0.1 will use the absolute criterion instead of the relative.

- **Sub-Problem Iterations**
 - **Maximum** - This is the maximum number of approximate problem cycles.
 - **Consecutive for Convergence** - The approximations will be repeated at least this number of times while satisfying the Sizing Level Convergence tolerance criteria.
- **Sequential Unconstrained Parameters**
 - **Maximum Number of Independent Pipe Sizes for Branch and Bound** - Not used when Sequential Unconstrained is directly selected.
 - **Number of Discrete Iterations** - This number of cycles with the artificial discrete constraints in place will be run. If this value is greater than 1, it is possible that discrete sizes more than one size away from the continuous solution can be found. However, this does not force an exhaustive search of all potential discrete sizes in this range.
 - **Continuous Penalties**
 - **Initial Multiplier** - This is the penalty applied to the Objective in the first continuous unconstrained cycle.
 - **Multiplier on Additional Iterations** - The penalty is increased by this amount each continuous cycle.
 - **Discrete Penalties**
 - **Initial Multiplier** - This is the penalty applied to the Objective in the first discrete unconstrained cycle.
 - **Multiplier on Additional Iterations** - The penalty is increased by this amount each discrete cycle.
- **Design Requirement Tolerances**
 - **Violated** - A very small positive value, which allows Design Requirements to be violated. It is important to allow this small violation for numerical reasons, as attempting to force values to zero is often difficult computationally.
 - **Active** - This small negative number determines how close to the Design Requirement the Objective can be before the Design Requirement influences the search direction. This value is updated automatically during the sizing process.
- **Scaling Parameters**
 - **Desired Objective Scaled Value** - In some cases it is helpful to scale the value of the Objective. Very large or very small unscaled Objective values can present numerical issues, and scaling can help with this.
 - **Scale Factor for Design Requirements**
 - **Common** - Design Requirements are always scaled (dividing by the user defined quantity) for numerical reasons. This factor allows additional scaling.
 - **Violated Tolerance** - This can be used to reduce the value of Violated Design Requirement Tolerance. This can be useful if the small violation is unacceptable for a feasible design.
- **Print Optimizer Debug File** - Generate a debug file detailing the normalized and generalized optimization process.

Related Topics

Search Methods.....	678
Modified Method of Feasible Directions.....	679
Sequential Quadratic Programming.....	681
Genetic Algorithm.....	686

Genetic Algorithm

A [Discrete Search Method](#) available in the [Sizing Method panel](#). This method is fundamentally discrete and does not use [gradients](#) in its search.

The Genetic Algorithm is a type of Evolutionary Algorithm - as the name implies, it selects advantageous traits by emulating biological evolution. An initial "population" of potential solutions is generated and gradually improved.

The starting population is an effectively random selection of "individuals". In the context of ANS, an individual is a proposed selection of pipe sizes for the system in question. At the start of the sizing process, these individuals may have an assortment of good and bad traits. A "bad" trait might be one that violates a Design Constraint or increases the cost of the system.

To improve the Fitness of the population there are three major operations: *Selection*, *Crossover*, and *Mutation*.

Selection occurs by determining the "Fitness" of all individuals in the population. Objective value is not the only concern when determining how acceptable any given individual is, so we need a different function to represent this. This function is a Fitness Function. The Fitness of any given individual is measured based on its Objective value and proximity to, or violation of, Design Requirements. Like Darwin's theory of *Survival of the Fittest* the goal in this process is to maximize the Fitness of the population as a whole. With the Fitness known for all individuals, a parent population with high Fitness can be selected.

Crossover occurs by "breeding" the parent solutions to generate new solutions. Some of these solutions will, randomly, have inherited good traits from their parents. Others will have inherited bad traits.

Mutation occurs after crossover, introducing a further element of randomness into the system by changing traits with no regard to the parent traits. This allows the algorithm to consider new solutions that were not previously part of the "gene pool." If no new traits can be introduced, the flexibility of the method is limited. If the mutation rate is too high, however, the good traits will not be carried through to new generations as effectively.

The new population is then combined with the parent population, and the process is repeated.

Note: When sizing with a Discrete method, it is possible to start the search with a Continuous Method. This limits the search space which should reduce the run time. When this option is enabled, the user has control over both the Continuous and Discrete methods.

Sizing Control Parameters

- **Genetic Algorithm Population Size**
 - **Number of Discrete Sizes Around Continuous Solution** - If Start with Continuous Method is selected, this option limits the search space of the genetic algorithm by only considering discrete sizes close to the continuous solution. For example, a Candidate Set may contain 20 unique sizes. Including all of these sizes makes the problem extremely complex, as indicated in [Brute Force Method](#). It is likely that the best discrete solution is near the best continuous solution, so only searching 4 unique sizes (+/- 2) around the continuous solution can dramatically reduce runtime.
 - **Automatic** - Set the **User Specified Multiplier** to 2 to determine the size of the population to use.
 - **User Specified Multiplier** - Every independently sized pipe in the model has a certain number of possible sizes that it may be, as determined by its particular Candidate Set. Because every independently sized pipe in the model can have a unique Candidate Set, it can also have a unique count of possible sizes. This multiplier is based on the sum of all of these counts across the model. For example, if there are 2 independently sized pipes with Candidate Sets P1: (4", 6") and P2 (3", 4", 6", 8"), the total count will be 6, and a multiplier of 3 would give a population size of 18.
 - **User Specified Population** - The size of the population can also be directly specified. It is important the population size is not too small as this will artificially limit the capability of finding a good solution. However, population sizes too large will not solve in a reasonable time frame.

Caution: The Advanced Control Parameters below should not be changed under normal circumstances without the direction of AFT Support.

- **Genetic Algorithm Seed** - This number is used to generate a "random" population. The same number will always generate the same initial population.
- **Genetic Algorithm Parameters**
 - **Crossover Probability** - The average fraction of the population obtained from crossover. As a probability this does not mean exactly this fraction will be obtained. This value is typically close to 1, meaning that the majority of the population each generation is obtained as children of the parent generation. It is generally less than 1 to avoid losing very good solutions from the parent generation due to poor crossover.
 - **Crossover Distribution Index** - This value is inversely proportional to allowed perturbations caused by crossover. That is, a large value will limit the amount of perturbation generated from crossover, whereas a small value will allow significant perturbation.
 - **Mutation Probability** - The average fraction of the population subjected to mutation. As a probability this does not mean exactly this fraction will be obtained. This value is typically the inverse of the number of independently sized pipes. As the system (and usually, population) gets very large, the need for "additional" randomness decreases. However, for small systems this randomness is important to introduce to prevent a premature solution.
 - **Mutation Distribution Index** - This value is inversely proportional to allowed perturbations

caused by mutation. That is, a large value will limit the amount of perturbation generated from mutation, whereas a small value will allow significant perturbation.

- **Genetic Algorithm Convergence**
 - **Consecutive Unimproved Generations** - When no improvements in the Objective occur over this number of generations, the sizing process will be stopped and the results are presented. It is assumed from the lack of progress that the ideal solution has been found. It is impossible to tell in advance how many total iterations this condition may require.
 - **Fixed Number of Generations** - To limit the runtime to a length that can be estimated, the algorithm can be set to a specific count of generations. This should be used with caution as it is possible to end before the best solution has been found, or to continue long after the best solution has been found.

Related Topics

Search Methods.....	678
Modified Method of Feasible Directions.....	679
Sequential Quadratic Programming.....	681
Sequential Unconstrained.....	683

Sizing Theory

Automated Pipe Sizing is a small subset of the mathematical field of Optimization. This is a complex subject that has evolved over many years - the topics here are not attempting to replicate this complexity but instead provide simplified understanding of the methods so that users of ANS can appropriately size systems and adjust models and parameters as needed.

- [Sizing and Optimization](#) - ANS is designed to simplify the optimization problem as much as possible. Those familiar with optimization may find this topic useful to connect common optimization terms to those used in ANS.
- [Boundaries vs. Design Requirements](#) - "Requirements" can be defined in many ways. Boundaries force a certain flow or pressure, whereas Design Requirements place limits on allowable values. While similar, the distinction is important in a sizing context.
- [Physical Parameters vs. Monetary Cost](#) - Monetary Cost sizing is often difficult to the amount of data required. For this reason, minimum cost is often estimated with a physical parameter, such as pipe weight.
- [Brute Force Method](#) - The most basic way to conclusively find a minimum is to exhaustively test all possibilities. This is impossible on a realistic time scale for most systems.
- [Gradient Based Methods](#) - The fastest methods for solving a continuous Objective are based on gradients. Understanding the core of these methods can help in finding better solutions more quickly.
- [Estimating the Gradient](#) - The gradient is only estimated - not analytically determined. This fact means the associated parameters may require adjustment in some cases. Understanding the affect of these adjustments is important.
- [Branch and Bound](#) - Only some methods are inherently suited to discrete sizing. Gradient based methods usually rely on narrowing the solution space and solving a number of easier problems.

Sizing and Optimization

Techniques for minimizing or maximizing certain parameters - optimization - have been used throughout a variety of industries for many decades. As computers become more prevalent and more powerful, they are able to handle larger problems with more variables and greater complexity. Optimization problems can be enormously complex, and techniques for solving them are almost always iterative. Such problems are generally impractical to solve without a computer.

The selection of pipe sizes to minimize some parameter - often monetary cost - is one specific example of optimization. For the sake of clarity, the mathematical terminology used when discussing optimization problems is compared to the terminology used in the ANS module here.

- **Optimization** - Modifying a system to minimize or maximize some parameter.
 - **In ANS** - This is represented by the name of the module - Automated Network **Sizing**. The sizes of the various network pipes are modified automatically by the application, to achieve some goal.
- **Objective** - The goal of optimization is referred to as the Objective. Minimizing monetary cost is a very common Objective. Others may be minimizing the time to complete an action, or maximizing the amount of material a machine can process.
 - **In ANS** - The term **Objective** is retained.
- **Objective Function** - It is required that the Objective can be calculated for any configuration of the system. This calculation is represented by the Objective Function, much like any other mathematical function. The Objective is to find the global maximum or minimum of this Objective Function.
 - **In ANS** - Is not a user input, and is not visible.
- **Constraint** - Anything that is required by the system regardless of the Objective value. For example, an Objective might be to minimize the amount of cloth needed to make a blanket. The smaller the blanket, the less material it needs, but a tiny blanket is not very useful. Instead, the Optimization might be Constrained such that the blanket must be some minimum area.
 - **In ANS** - These are called **Design Requirements**.
- **Design Variable** - Anything that is allowed to change in order to modify the Objective value. In the blanket example, this might be the width, length, and thread count of the blanket. As with any mathematical problem, the more variables there are, the harder the problem is to solve.
 - **In ANS** - The variables that are allowed to change are the pipe sizes. There are countless ways a fluid system could be Optimized, but pipe sizing is one of the most significant that will impact cost. The total number of design variables in the current analysis is displayed near the bottom of the [Size/Cost Assignments panel](#) as **Independently Sized Pipes**.

Related Topics

Sizing Theory.....	688
Boundaries vs. Design Requirements.....	690
Physical Parameters vs. Monetary Cost.....	691
Brute Force Method.....	692
Gradient Based Methods.....	693

Estimating the Gradient.....	697
Branch and Bound.....	701

Boundaries vs. Design Requirements

Boundaries

Points in a model that **physically force a given parameter**.

A piping network can be considered to have three variables - *flow*, *pressure*, and the *system*. For analysis, it is required that the user fully define the *system*. The *system* is a collection of components that affect the *flows* and *pressures* - pipes, pumps, valves, etc. Because there are still two variables - *flow* and *pressure* - the user must specify at least one of them as a boundary somewhere in the system.

Note: Due to the nature of fluid analysis, if there is only one boundary it must be a pressure type boundary. See [Role of Pressure Junctions](#).

For a model to be accurate, its *entire* definition must reflect the system it is representing. This includes both the *system* and the *boundaries*. It is obvious that a 120 foot, 12" diameter pipe should be represented in the model with the same length and diameter. However, ensuring the correct selection of boundaries is often less intuitive.

Specifying any given boundary as a fixed pressure will result in some calculated flow at that boundary. Changing this pressure boundary to a flow boundary with the calculated value will not change any of the pressure or flow results in the model. However, this has important physical implications.

For example, consider a pipe bounded by two pressures. Doubling the pipe size will significantly increase flow. However, if a boundary was instead a fixed flow, doubling the pipe size will *not* change the resulting flow but instead change the pressures at the flow boundary. If the boundary in the real system is a large fixed pressure tank, it should not be represented as a fixed flow, because changes to the system will effectively change the pressure required in the tank. In other words, **changing the system changes the physical meaning of the boundary**. If the tank is represented as a fixed pressure, as it behaves in the field, this is not an issue.

This is important in ANS because ANS is **changing the system** as part of the sizing process. Using the incorrect type of boundary can present sizing solutions that would not work in the real system.

Note: It is not uncommon or incorrect to use boundaries to represent a *requirement* in an analysis. This is a common design tool to determine what is required of a boundary. In the above example, it may be desired to know the required tank pressure to drive the required flow. What is important is to recognize whether a junction is representing a requirement or a physical boundary that cannot be changed.

Design Requirements

Design Requirements are **limits on allowable behavior**.

The distinction is important because Design Requirements are *limits* rather than *imposed* values. For example, a Design Requirement may state that a flow must be **at least** 100 gal/min. The design is

considered acceptable if the flow ends up as 120 gal/min. On the other hand, using a boundary of 100 gal/min **forces the model always to maintain 100 gal/min**.

In this manner, Design Requirements are *less* restrictive than boundary conditions. Generally, the less restrictive the problem is, the more flexibility is available to obtain a better sizing result.

Even if it is known that there must be some fixed flow into a tank, using an Assigned Flow boundary at this location may not be the best choice. If the pressure in the tank cannot be changed to the value calculated based on the sizing result, the use of an Assigned Flow here would be incorrect. Instead, it would be appropriate to use an Assigned Pressure and two Design Requirements on flow - a minimum and a maximum, to limit the solution to an acceptable range.

Related Topics

Sizing Theory.....	688
Sizing and Optimization.....	689
Physical Parameters vs. Monetary Cost.....	691
Brute Force Method.....	692
Gradient Based Methods.....	693
Estimating the Gradient.....	697
Branch and Bound.....	701

Physical Parameters vs. Monetary Cost

There are four physical parameters that can be maximized or minimized in addition to monetary cost:

- **Pipe Weight**
- **Pipe and Fluid Weight**
- **Flow Volume**
- **Pipe Inner Surface Area**

Actual Objective is to Limit the Physical Parameter

Occasionally, the goal of the sizing is to limit the physical parameter. For example, a system to be placed on an aircraft likely has greater emphasis on weight than monetary cost. It is important to keep in mind in such cases that **only the pipes are included in the total Objective value**. Pumps, valves, or other junctions that contribute to weight are not accounted for. This may affect the ideal sizing.

Using a Physical Parameter to Estimate Cost

There are advantages and disadvantages of estimating cost with a physical parameter.

It is generally far easier to use a physical parameter as **cost libraries are not required**. Cost libraries are difficult to locate information for, and can be time consuming to create and manage. Avoiding this greatly simplifies the process.

The main disadvantage is that **only initial pipe costs can be accounted for**. If junctions account for a significant portion of the initial cost, or if the recurring costs from operating them would affect sizing, then this may not result in the best sizing solution.

In general, if the goal is to minimize monetary cost of a system, then monetary information should be supplied. If recurring costs must be included, this is the only reasonable option. However, physical parameters often provide a good estimate of the ideal size for initial cost. It makes sense to start the sizing process with a physical parameter, and proceed to monetary cost as the Objective after larger design issues have been resolved. This is especially true if the potential savings are large.

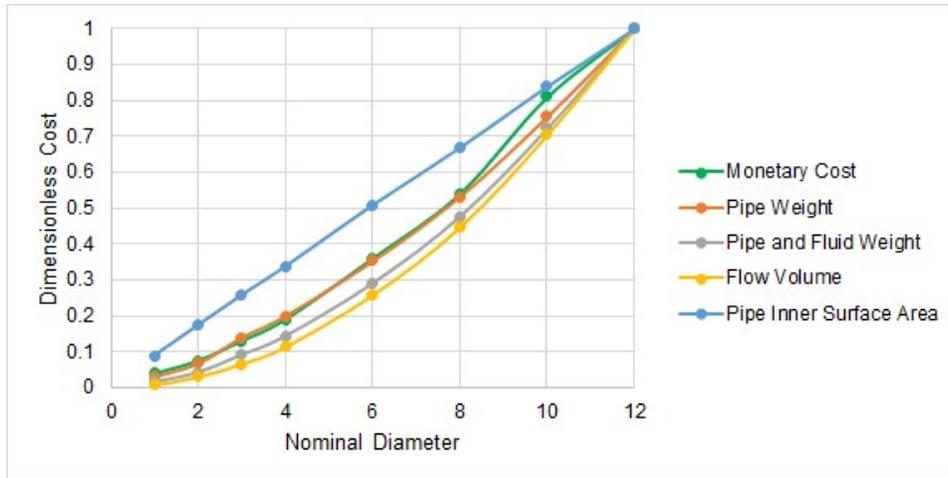


Figure 1: Comparison of Dimensionless Costs for Schedule 40 ANSI Steel water pipe. RS Means monetary cost closely follows pipe weight in this example.

Related Topics

Sizing Theory.....	688
Sizing and Optimization.....	689
Boundaries vs. Design Requirements.....	690
Brute Force Method.....	692
Gradient Based Methods.....	693
Estimating the Gradient.....	697
Branch and Bound.....	701

Brute Force Method

The "easiest" way to find the cheapest set of pipe sizes is to test all of the possible combinations exhaustively, and report back the lowest cost option found.

This has the advantage that it is clearly conclusive - there can be no argument that a cheaper alternative exists if every possible alternative was shown to have higher cost. Unfortunately, this method is extremely impractical for any reasonable sized problem. The number of required analyses to find the current value of the Objective is related to the number of design variables and how many unique discrete sizes each variable can have.

In the context of ANS, this is the number of Independently Sized Pipes, and the number of selections in the Candidate Set, assuming there is one set for the model.

$$\text{Unique Hydraulic Models} = (\text{Candidate Set Count})^{\text{Number of Independently Sized Pipes}}$$

If we have only 3 independently sized pipes, and can only choose from 3 different pipe sizes, then we would need to solve 27 unique hydraulic models, which could be manageable by hand.

However, if we have 12 independently sized pipes, and can choose from 6 different pipe sizes, we suddenly have over **2 billion** unique models to solve. If we could test 50 models every second, this would take over a year to complete - and it is still a relatively small system!

Due to the clear impracticality of this method, it is not used in ANS.

Related Topics

Sizing Theory.....	688
Sizing and Optimization.....	689
Boundaries vs. Design Requirements.....	690
Physical Parameters vs. Monetary Cost.....	691
Gradient Based Methods.....	693
Estimating the Gradient.....	697
Branch and Bound.....	701

Gradient Based Methods

Gradient based methods search for the ideal solution using the **gradient** of the Objective Function. A gradient is a vector function composed of partial derivatives measuring the change of some function with respect to each component. Effectively, a gradient is a derivative of a multi-variable function - **it indicates the magnitude and direction of greatest change** in a multi-variable function.

A gradient of a single variable function is simply the derivative of that function. While most problems requiring complex optimization techniques have many variables, it is easiest to think about a single-variable function, such as elevation related only to distance along some horizontal line:

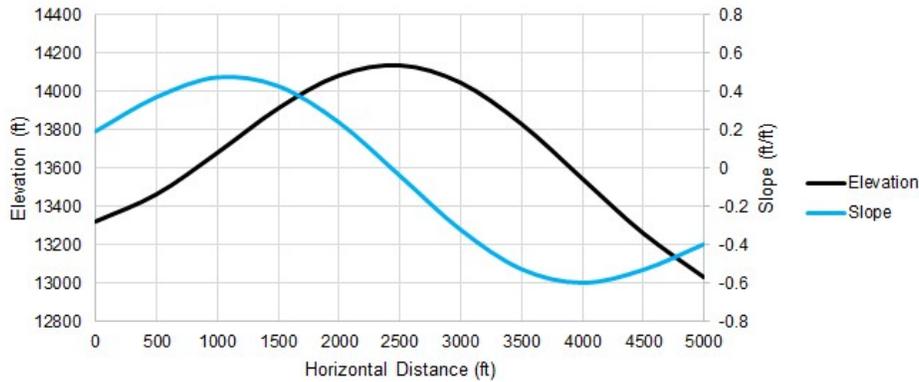


Figure 1: One of many possible elevation profiles of a mountain peak. The slope can be calculated at any point.

The gradient of a mountain's elevation will indicate the magnitude and direction of the elevation change at any given latitude and longitude. In other words, given a specific point, the gradient will tell you how steep the slope is at that point, and which direction it is steepest in. This is very useful for finding the top of the mountain - without being able to see anything but the local value of the gradient, you can make an estimate that the peak is probably in the direction of the slope, and that if the slope is very steep you likely have a long ways to go.

If the entire elevation profile was known ahead of time, finding the peak is trivial. Unfortunately, we do not normally know the entire profile, but instead can only determine its value where we explicitly test it. Stated generally, **the shape and behavior of an Objective Function is not known ahead of time.**

Rather than testing every single horizontal point to find the highest elevation by [brute force](#), we can employ an intelligent gradient based search method. Starting at a horizontal distance of zero, we can measure an elevation and slope:

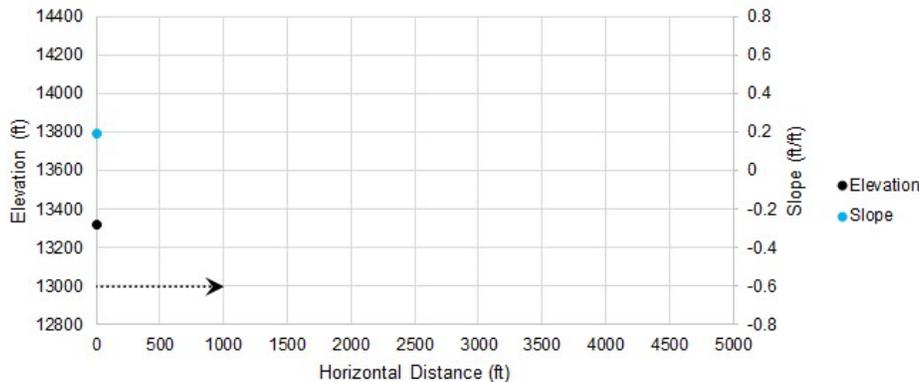


Figure 2: Starting a gradient based search

With the positive slope from our initial condition, we can make an educated guess that the peak is somewhere in the positive direction. We determine where we want to guess and measure our system again at that point:

Cost Calculation

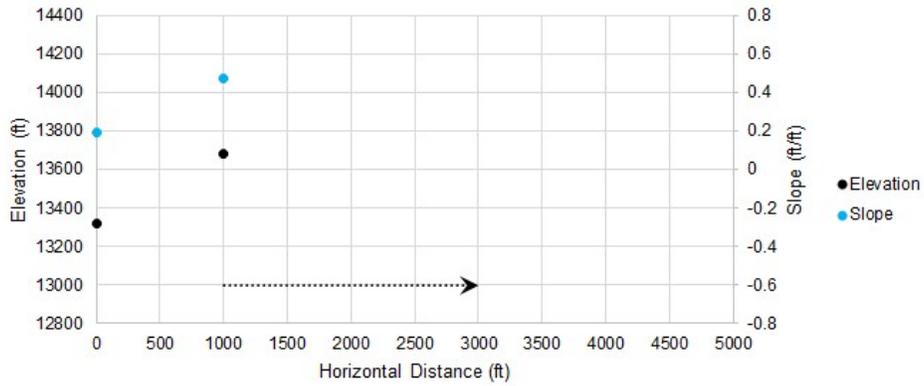


Figure 3: Iteration number 2

Because our slope is higher, we assume that the peak is farther away and adjust our change in distance accordingly. Our new guess has a negative slope, so we will reverse our search direction.

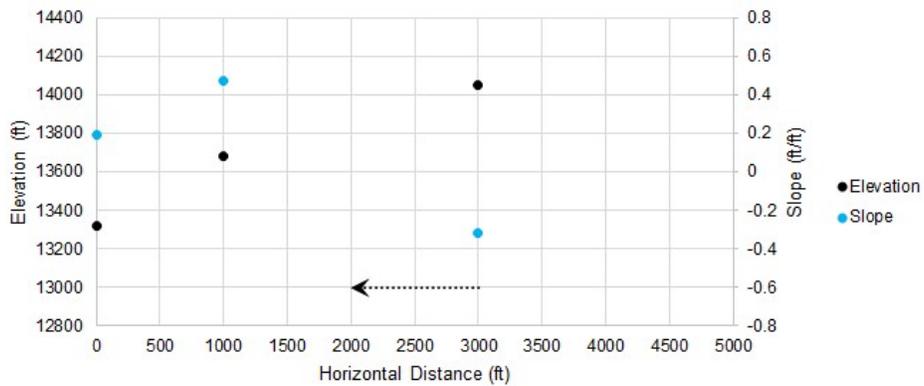


Figure 4: Iteration number 3

We changed sign on slope again, so we reverse direction again. We can use information from previous guesses to narrow down the search range and make a more intelligent guess.

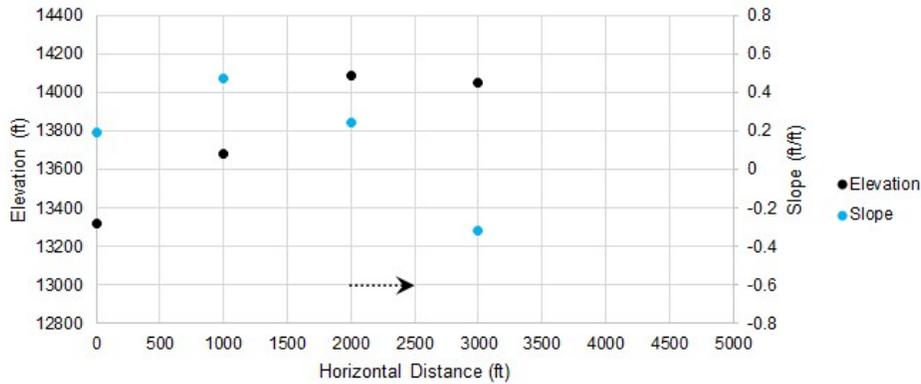


Figure 5: Iteration number 4

As we continue, the adjustments to our guess get smaller and smaller. Once they are small enough, we can consider the search converged, and the peak found.

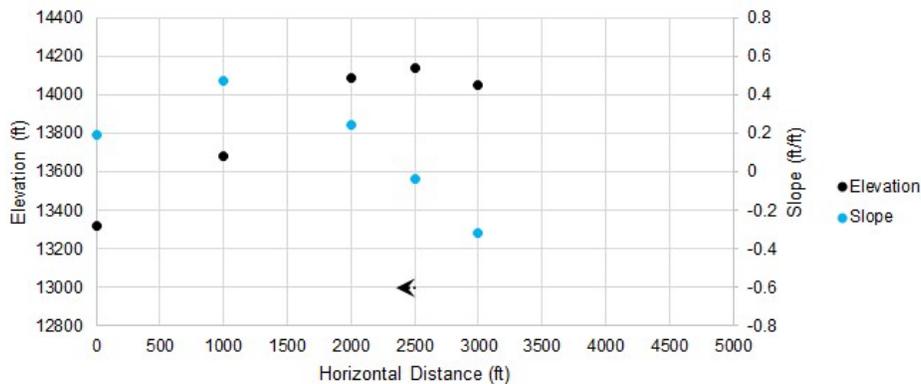


Figure 6: Iteration number 5

In only a few iterations, we have found an excellent approximation of the highest point, without knowing anything about the elevation or slope outside of our selected guess points.

The [search algorithms](#) included in ANS are much more sophisticated than the simple example presented here, but the idea remains the same. Furthermore, the allowable selections are restricted by [Design Requirements](#), which impacts the ideal search direction.

Related Topics

Sizing Theory.....	688
Sizing and Optimization.....	689
Boundaries vs. Design Requirements.....	690
Physical Parameters vs. Monetary Cost.....	691
Brute Force Method.....	692

Estimating the Gradient.....	697
Branch and Bound.....	701

Estimating the Gradient

Analytically determining the gradient becomes extremely difficult if not impossible for problems with many variables such as sizing a piping network. Instead, the gradient is estimated with a **finite difference**. This is a method for numerically estimating the gradient at a given point.

The test points created with a finite difference method each represent a hydraulic model that must be solved independently. If the model is complex, adding additional variables (unique pipe sizes) or required runs (from a central difference) can make the process take a significant amount of time.

Forward Difference

The Forward Difference estimates the gradient by finding the change in the function value by making a small change in the independent variable.

First, an initial result is found. We want to know the gradient at this point.

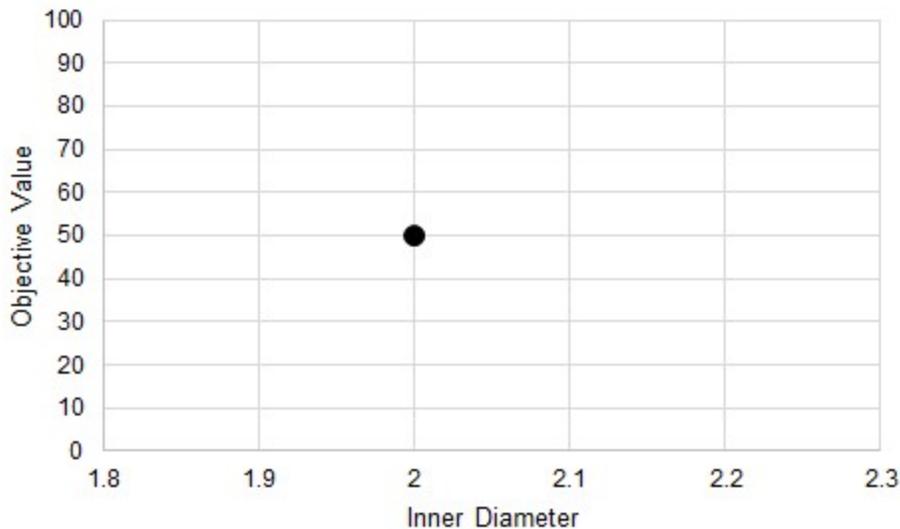


Figure 1: An initial test. It is desired to know the gradient at this point.

The independent variable is perturbed some small amount and a new result found. A line can be drawn through these two points - an estimate of the gradient at the initial point.

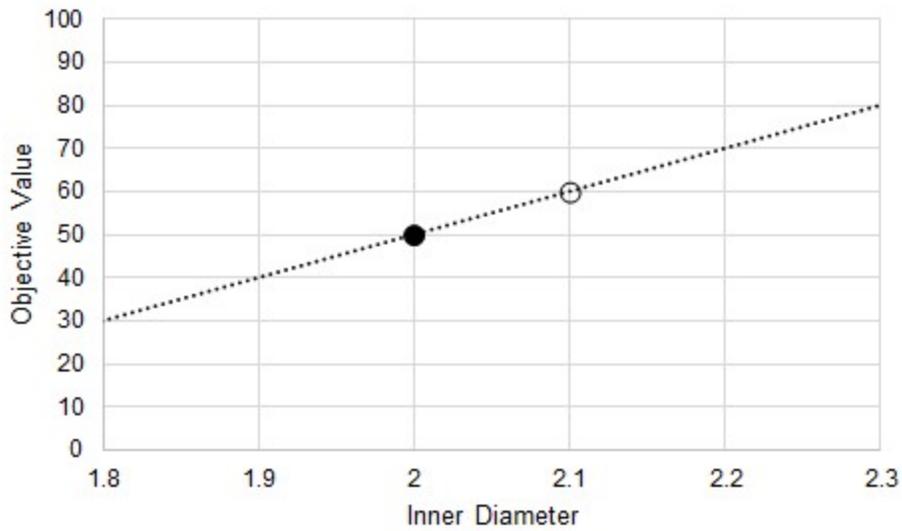


Figure 2: A perturbation. This allows the calculation of an average gradient as an approximation of the gradient at the initial point.

There are two important things to keep in mind:

- This is an average gradient between the two points, but it is **used as an estimate of the gradient at the initial point**. It is not used to represent the behavior between the two points.
- It is only an estimate. **The result depends on the behavior of the Objective Function, and the selection of the test point.**

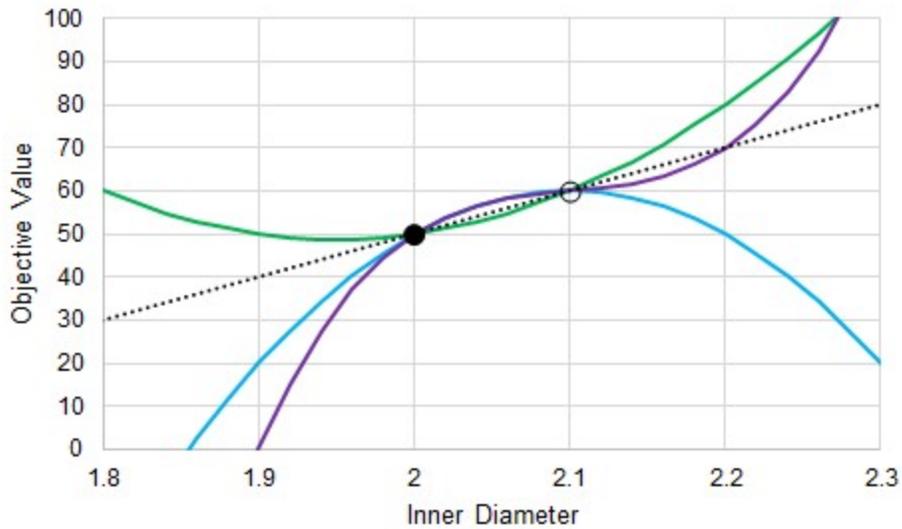


Figure 3: The actual gradient depends on the function. Many possible functions pass through two test points.

Central Difference

The central difference is a very similar process, with generally increased accuracy. We still want to know the gradient at our initial point (Figure 1), but instead of creating one test point, we create two, equidistant from the initial position. This effectively doubles the number of calculations required.

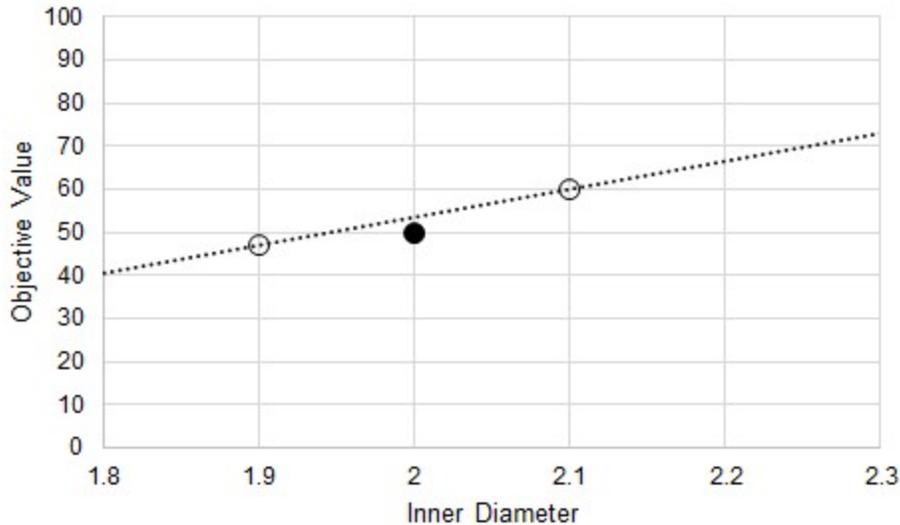


Figure 4: Central Difference estimate of gradient at the initial point

Relationship to Iterative Tolerance

It may seem that simply making the perturbation in the Inner Diameter smaller will increase the accuracy of the gradient and help reduce the issue indicated by Figure 3. If the Objective function was known explicitly, this would be a good assumption. However, in ANS, the results are not known explicitly. Instead, the solution is found via [iteration](#). Iterative methods require some determination of convergence - namely, a *tolerance* on the solution. This means, by definition, that the results found are not the exact, analytical solution. Further, it is impossible to tell how the results may be biased away from the true solution.

When making test points, as before, there is some band of uncertainty in the Objective Value for the point related (but not necessarily equivalent) to the iterative tolerance.

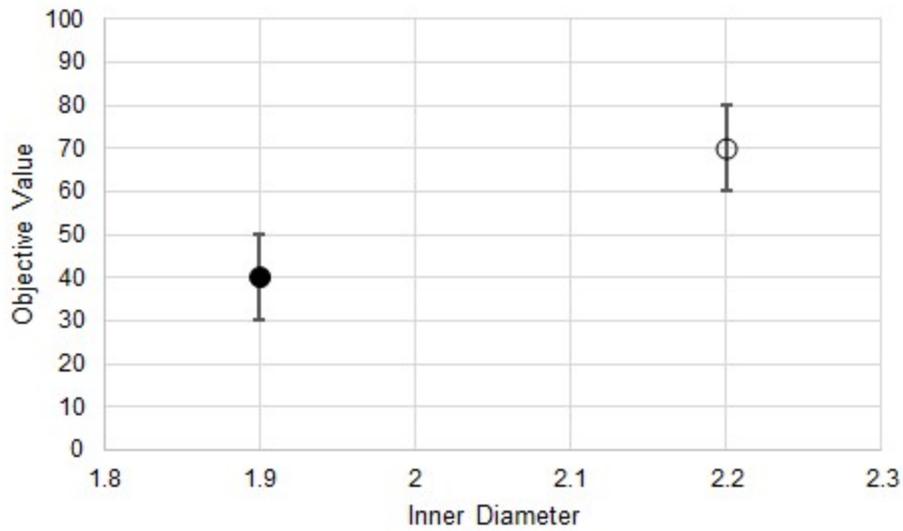


Figure 5: Uncertainty in the Objective Value due to iterative tolerance

We do not know where the "true" Objective Value lands on either of these bands, and we cannot state that the error is in the same direction or is of the same magnitude for both test points. This means that the gradient estimations have some variability as well.

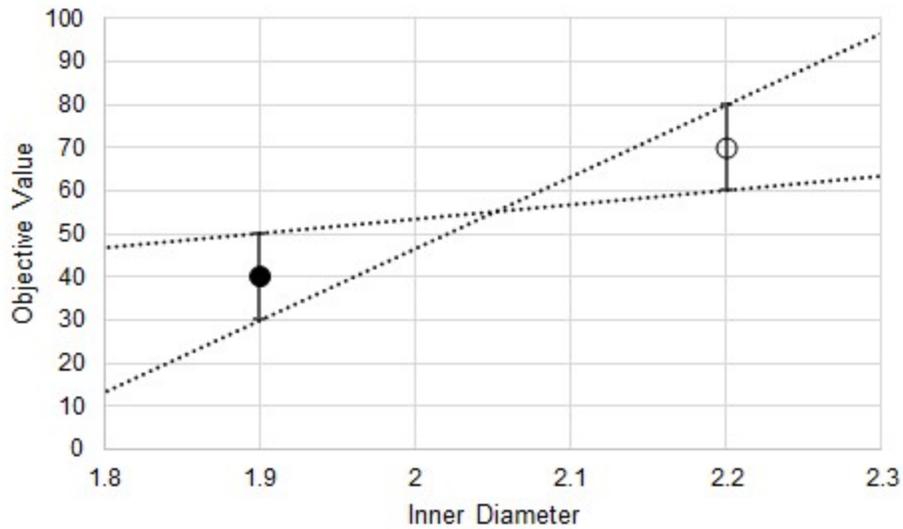


Figure 6: Uncertainty in the estimated gradient

In addition to the uncertainty indicated by Figure 3, there is also added uncertainty from the iterative tolerance.

It can be tempting to reduce the step size to increase gradient accuracy, but the iterative error will not be reduced. This can have dramatic results on the gradient.

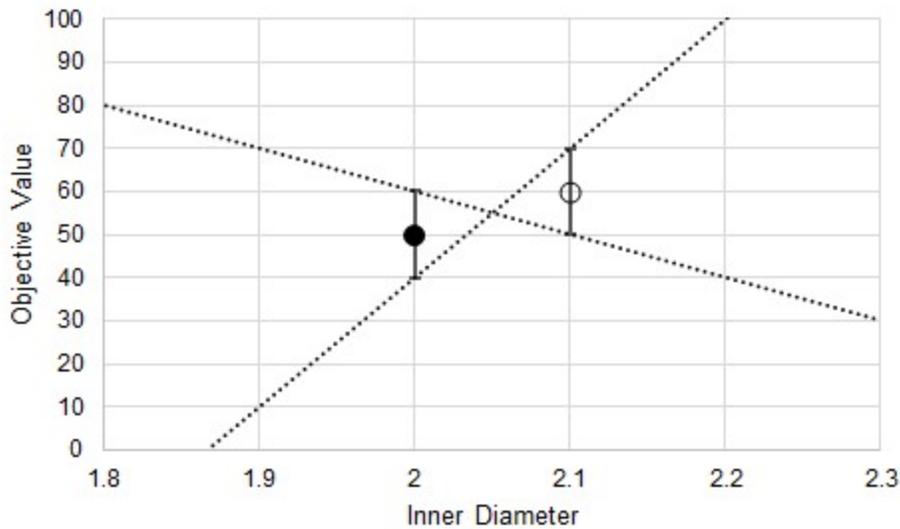


Figure 7: Substantially increased uncertainty in the estimated gradient from decreasing the step size

Now, due to the iterative tolerance, it is possible that the gradient is dramatically affected to the point where even the sign is uncertain.

Note: Making the step sizes too large impacts the gradient negatively in other ways. The critical thing to remember is that the step size should be small enough to capture the changing Objective, but not so small the iterative tolerance impacts the gradient quality.

Related Topics

Sizing Theory.....	688
Sizing and Optimization.....	689
Boundaries vs. Design Requirements.....	690
Physical Parameters vs. Monetary Cost.....	691
Brute Force Method.....	692
Gradient Based Methods.....	693
Branch and Bound.....	701

Branch and Bound

Piping is almost always only available in [Discrete](#) sizes. Determining the best case selection of Discrete sizes requires a different approach than finding the best case Continuous sizes. One such method is **Branch and Bound** which is used for Discrete sizing on the basis of one of the Gradient Based Methods.

Branch and Bound divides the sizing problem into a collection of smaller sizing problems (branches). Solving these "sub-problems" gives information (bounds) about whether or not a given sub-problem is worth further investigation.

This method is well suited to problems with a relatively small number of variables (unique pipe sizes). For more than 10 variables, the [Sequential Unconstrained](#) method is advised. It is difficult to tell in advance how well any search method will work, and Branch and Bound is no different. It is possible, though unlikely, that the method can effectively reduce to a Brute Force Search. This makes it effective, but potentially very slow.

Example

There is a 50 foot pipe followed by a 100 foot pipe, and a 5 psi differential across the pipes. The piping is available in integer inside diameters between 3 and 17 inches, and the pipe walls thickness is always 10% of the inside diameter. Our goal is to minimize the weight of this system, while maintaining a flow of at least 2500 gal/min.

Even though there are only two independent pipe sizes to consider, with 15 possible pipe sizes, there are 225 possible permutations, more than we would like to test by brute force.

We start the process by finding the Continuous Optimum. For this example, this best case is (P1 = 8.75 in, P2 = 8.77 in, Weight = 2772 lbm). It is reasonable to assume that the Discrete Optimum will be near these sizes.

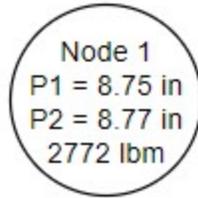


Figure 1: The first node showing the Continuous Optimum

From this first **node**, we want to divide the solution space on our first variable - the diameter of pipe 1. The pipe size cannot be 8.75 inches, it can only be a discrete size, in our case an integer value. Therefore, we **branch** the into two sub-problems with pipe 1 restricted to less than 8 or more than 9 inches (Figure 2). We have also established a **bound** on the problem - the solution will never have a feasible weight lower than 2772 lbm.

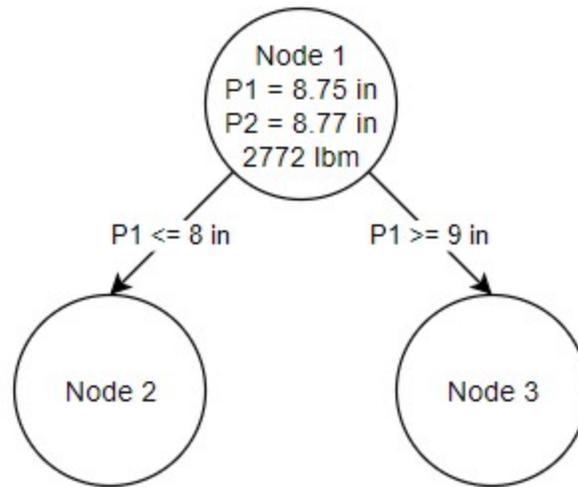


Figure 2: Two Sub problems are created from the Continuous Optimum in Node 1

It is important to note that these additional nodes represent **continuous sub-problems**. The pipe size is not forced to be discrete, it only has an additional requirement on it. The resulting pipe size will often be discrete because that continuous diameter is closest to the overall lowest feasible weight.

We can solve each of these independent problems with the same methods we used to solve the first node.

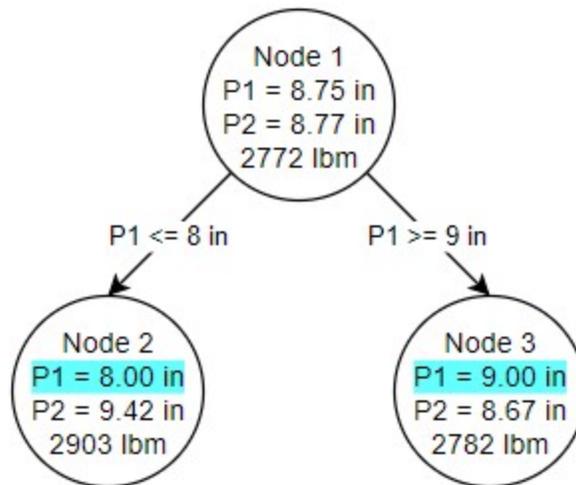


Figure 3: The conditions of Nodes 2 and 3 are shown, as well as the sizing arrived at in the two nodes

Due to the nature of this particular problem, the ideal pipe sizes for pipe 1 in both nodes 2 and 3 ended up as discrete sizes. This is not required or enforced by the method. Each of our nodes also has its own bound. This fact is used to determine the location of the next branch. Because node 3 has a better pos-

sible solution (2782 lbm) than node 2 (2903 lbm), we branch from node 3 (Figure 4). This time, however, we place a restriction on our second pipe's diameter.

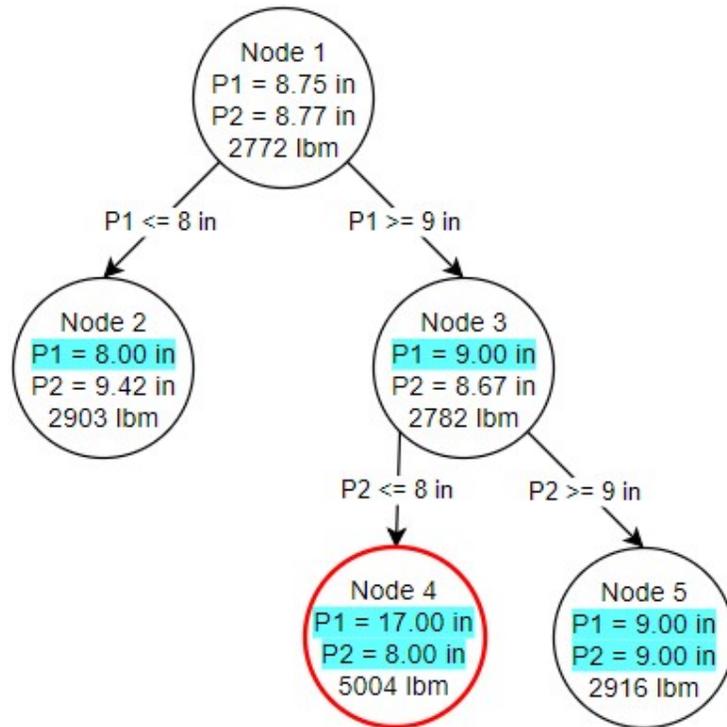


Figure 4: Node 4 produces an infeasible solution while Node 5 produces a feasible solution based on the branching criteria set from Node 3

Both of our branches from node 3 resulted in discrete size selections. However, node 4 was not able to result in a feasible solution. **Infeasible nodes are valuable as there is no need to branch from them - placing further restrictions on a problem will never make it feasible.** Node 5 on the other hand is both fully discrete and feasible. This is our first **potential discrete solution**. It is only a potential solution because node 2 has a lower bound - without branching from node 2, we do not know if a better discrete solution may exist under it, so branching from node 2 is exactly how we proceed (Figure 5).

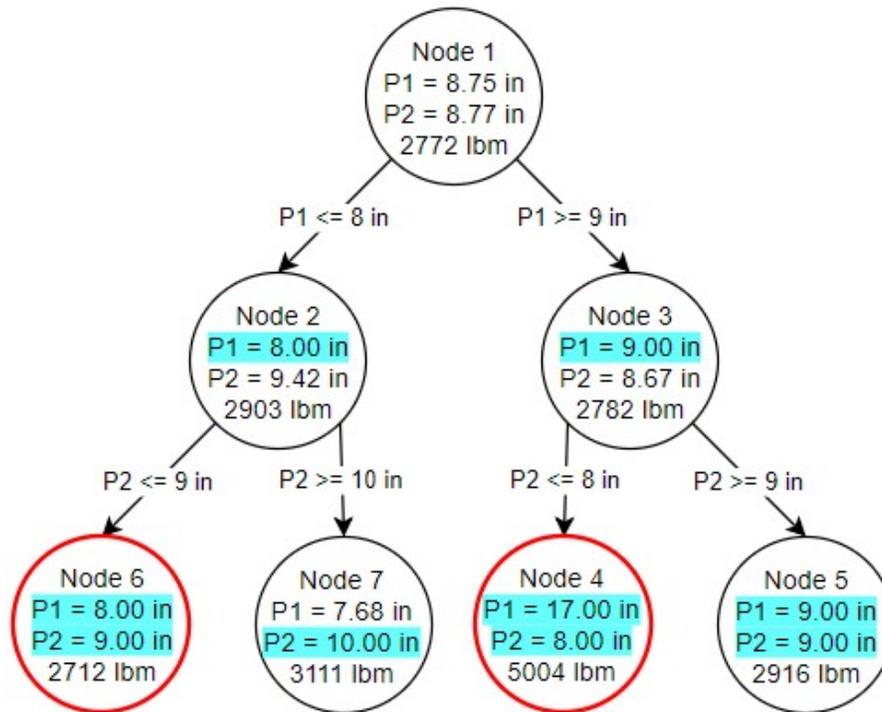


Figure 5: Branching occurs from Node 2 which produces one feasible (Node 7) and one infeasible (Node 6) solution

One of our nodes is infeasible, and despite the lower weight cannot be considered. Node 7, however, is feasible but not discrete, despite the requirements on diameter on both pipes. We could branch again from node 7, but there is no need - **we have already found our ideal solution at node 5**. We know this is the best possible discrete solution because the node 7 continuous solution has a higher minimum weight than node 5. No matter how we branch from node 7, we will never get a better solution than node 5.

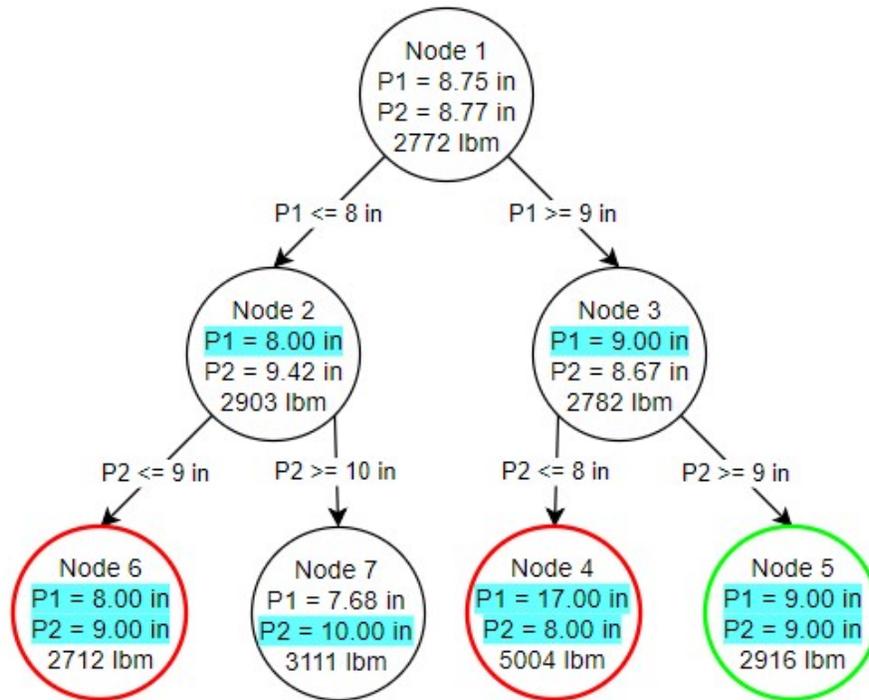


Figure 6: Results of the Branch and Bound method

Again, note that the results on node 5 are a **continuous** solution. It is only because of the requirements we have placed on the problem that the sizes have **resulted** in discrete sizes.

We have managed to find the best possible discrete solution by running 7 independent continuous sub-problems, instead of hydraulically solving hundreds of possible configurations. With more independent pipe sizes, the elimination of certain branches often becomes more common, further reducing the cases we need to check.

Related Topics

Sizing Theory.....	688
Sizing and Optimization.....	689
Boundaries vs. Design Requirements.....	690
Physical Parameters vs. Monetary Cost.....	691
Brute Force Method.....	692
Gradient Based Methods.....	693
Estimating the Gradient.....	697

Libraries

Powerful customization capabilities are provided through the ability to create libraries which can store custom fluid properties, pipe materials, and all junction types except the [Volume Balance](#). There are several categories of libraries which are discussed below.

AFT Internal Library

- **Purpose** - Stores AFT-provided library data that is always available with the default installation.
- **Filename** - Fathom#.dat (where # is the corresponding version number).
- **Location** - Fathom installation folder, default is C:\AFT Products\AFT Fathom #.
- **Limitations** - Cannot be transferred to other users, local use only.

Local User Library

- **Purpose** - Stores user-specified data for most component types.
- **Filename** - Fth_User#.DAT (where # is the corresponding version number).
- **Location** - C:\Users\username\AppData\Roaming\Applied Flow Technology\AFT Fathom\#.
- **Limitations** - Cannot be transferred to other users, local use only.

External Library

- **Purpose** - Stores user-specified data of any type: fluids, pipe materials, junctions, and fittings & losses. Multiple network or local custom libraries can be created via the [Library Manager](#) on the [Library menu](#). External libraries can be shared to multiple users from a network location as is discussed in [Creating an Enterprise-wide Network Library System](#).
- **Filename** - User-specified filename.
- **Location** - User-specified location.
- **Limitations** - Must be connected via the Library Manager prior to use, and consideration should be given to version control or user permissions to edit.

Related Topics

Library Manager.....	708
Library Browser.....	708
Network Library Overview.....	723
Library Verification.....	727
Library Locations in General Preferences.....	729
Fluid Library.....	713
Pipe Material Library.....	714

Pipe Material Library References.....	766
Junction Library.....	712
Insulation Library.....	717
Fittings & Losses Library.....	717

Related Examples

Hot Water System.....	950
Cooling System - Creating Libraries - ANS.....	1157

Library Manager

Accessed from the [Library Menu](#), the Library Manager allows you to:

- Connect to and disconnect from libraries
- Create and delete libraries
- Add content to libraries
- Review the content of libraries
- Move contents between libraries

Related Topics

Library Browser.....	708
Fluid Library.....	713
Pipe Material Library.....	714
Junction Library.....	712
Insulation Library.....	717
Fittings & Losses Library.....	717
Libraries.....	707
Network Library Overview.....	723
Library Verification.....	727
Library Locations in General Preferences.....	729
Pipe Material Library References.....	766

Related Examples

Cooling System - Creating Libraries - ANS.....	1157
--	------

Library Browser

The first panel of the [Library Manager](#) is the Library Browser. Use the Library Browser to create a new library, add an existing library, view contents of an added library, and connect or disconnect libraries to the model.

Types of Libraries

There are three types of external libraries in AFT Fathom:

- [Engineering Libraries](#) - Contain all component types except cost information
- [Energy Cost Libraries](#) - Contains energy cost data associated with components in existing engineering libraries
- [Cost Libraries](#) - Contains material, installation, and/or maintenance cost data associated with components in existing engineering libraries

All library types can be reviewed, added to the model, and connected to the model from the Library Browser. For engineering-type libraries additional actions are available to [edit libraries](#).

Adding and Removing Existing Libraries

To work with a library you must first make it available to AFT Fathom. Referring to Figure 1, clicking on the Add Existing Library or Add Existing Cost Library button will open a standard file dialog box with which you may browse to and select a library. A library may be removed from the list of available libraries by right-clicking on the library and selecting Remove Library from List. A summary of the library contents can be seen in the section on the right when a library, library section, or component is selected in the list of available libraries.

The AFT DEFAULT INTERNAL LIBRARY and the AFT FATHOM [LOCAL USER LIBRARY](#) may not be removed and will always appear in the list of available libraries. External shared libraries will also always appear in the available library list.

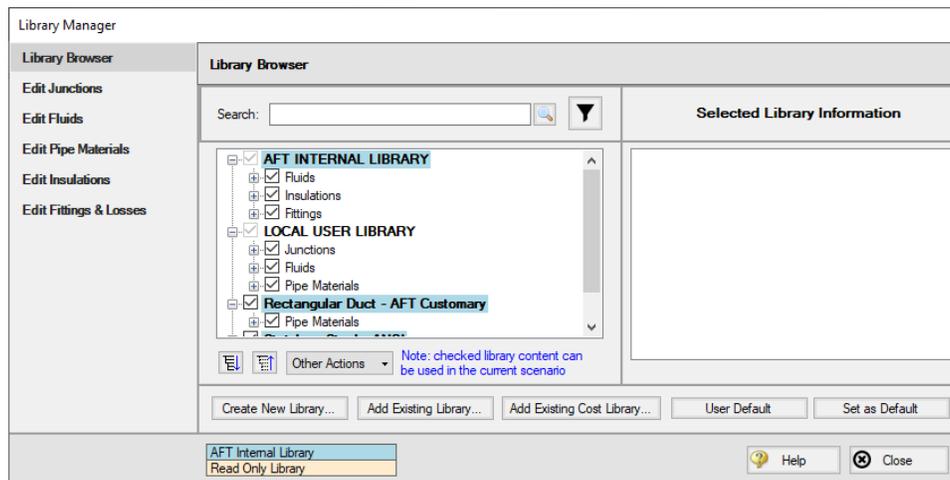


Figure 1: Library Browser in the Library Manager

Connecting and Disconnecting Libraries

In order to use the components from a library in the model the library must be **connected**. A library is connected to the model when it is shown with a check mark in the Library Browser. All libraries are automatically connected when they are first added to the Library Manager.

Sections or components in a library can also be connected/disconnected on an individual basis if the library as a whole is connected.

It may be desired to disconnect a library if you do not want the components from that library to be available for use in the current model, but you do want the library to be available for use in other models.

Clicking on Set As Default will establish the list of connected libraries as the default list, while clicking on User Default will set the connections to the default list.

AFT Internal Pipe Material libraries can be connected or disconnected as a group from the Other Actions button under the available libraries list.

Editing Library Contents

Existing content can be moved to/from, copied to/from, or deleted from available libraries in the Library Browser.

To edit an item right-click on the item in the Library Browser, then click the desired action from the right-click menu. Note that moving, copying, and deleting can be done to a library as a whole, or can be done for just a section or single component in a library. Individual components can also be edited by choosing Edit Component, which will take you directly to the Edit section for the component type.

Note: Editing Cost libraries must be done separately in the [Cost Library](#) or [Energy Cost Library](#) windows, accessed from the Library Menu.

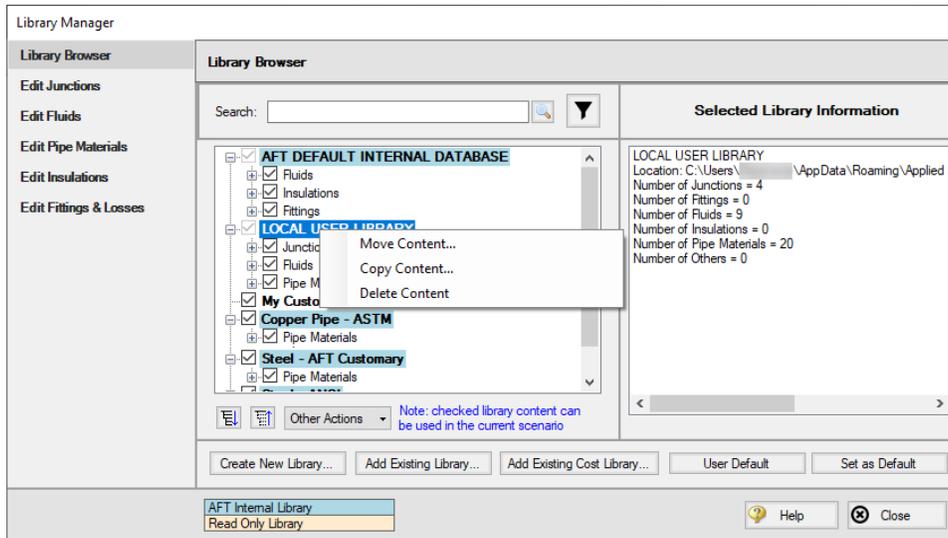


Figure 2: Library contents can be transferred or deleted from the right-click menu

Creating External Libraries

By default library items are stored in the AFT Default Internal Library or Local User Library, both of which cannot be shared with other users. Library items can be moved or copied to an external library which can be shared with multiple users.

To create an external library click the Create New Library button underneath the available libraries list. You will be prompted to provide a name for the file, and a library description (Figure 3). The provided description will be displayed as the library's name in the Library Manager.

To add content to an external library copy or move content from an existing library as described in the Editing Library Contents section. Junction components can also be directly added to external libraries as is described in the [Junction Libraries](#) topic.

For more details on creating and sharing external libraries see [Creating an Enterprise-wide Network Library](#).

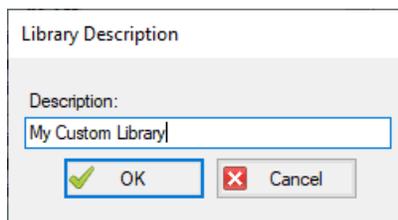


Figure 3: After specifying a library file name, you will be prompted for a description which will then appear in the Library Manager when this library is selected

Related Topics

Library Manager.....	708
Fluid Library.....	713
Pipe Material Library.....	714
Junction Library.....	712
Insulation Library.....	717
Fittings & Losses Library.....	717
Libraries.....	707
Network Library Overview.....	723
Library Verification.....	727
Library Locations in General Preferences.....	729
Pipe Material Library References.....	766

Junction Library

Adding Custom Junction Data to the Component Library

To simplify modeling and eliminate repetitive data entry, AFT Fathom allows you to save frequently-used junction data to a custom library. Once saved, the custom equipment data can be selected from the Library Jct dropdown in the [Properties window](#) at any later time for immediate recall of the data. Using this tool, you can easily build your own library of pumps, valves, or other components supplied by specific manufacturers.

To save a custom junction to the library - open the junction Properties window and enter all required data. Close the window and choose Add Component to Library from the [Library menu](#). You will be prompted to enter a name for the custom junction. Enter a descriptive name then click OK. If you open the junction again, the custom name will be selected in the Library Jct dropdown.

AFT Fathom accepts custom data for all junction types except the Volume Balance.

Editing the Component Library

You can delete or rename any existing library component in the Edit Junctions panel of the Library Manager, accessed from the Library menu. To edit an existing library entry, select the item in the list and click the Edit Data button. This opens the data into a Properties Window where the data can be modified.

Related Topics

Library Manager.....	708
Library Browser.....	708
Fluid Library.....	713
Pipe Material Library.....	714
Insulation Library.....	717
Fittings & Losses Library.....	717
Libraries.....	707
Network Library Overview.....	723

Library Verification	727
Library Locations in General Preferences	729
Pipe Material Library References	766

Related Examples

Hot Water System	950
------------------------	-----

Fluid Library

AFT Fathom provides an [AFT Standard fluid library](#) for numerous common fluids. If the working fluid you wish to model is not in this library, you may elect to enter the fluid properties in a custom library for later reuse using the Edit Fluids panel in the [Library Manager](#).

You can open the Edit Fluids panel from the Library menu or from the Edit Fluid list button in the [Fluid panel](#) in Analysis Setup. The Edit Fluids panel allows you to review currently defined AFT Standard and custom fluids, to change the data for custom fluids, and to enter new fluids. All AFT Standard fluid properties are assumed to be a function of temperature. Custom fluids can be a function of temperature or solids concentration.

Once you have entered a custom fluid, it will appear in the Fluids Available in Library list of the Fluid panel when User Library Fluid is selected.

Adding a Custom Fluid

You can add new fluids by clicking the Add New Fluid button on the Edit Fluids panel of the Library Manager. Raw data can be provided for the variation of density, dynamic viscosity, specific heat, thermal conductivity, and vapor pressure as they depend on temperature or solids concentration. This window provides the tools to perform polynomial curve fits of fluid data, or the raw data can be interpolated.

Required Data:

- Density
- Dynamic Viscosity

Optional Data:

- Specific Heat (required for heat transfer capabilities)
- Thermal Conductivity (required for heat transfer capabilities)
- Reference Temperature/Enthalpy point (required for heat transfer capabilities) - Note that the reference temperature/enthalpy do not need to be exact values since only changes in the parameters are used
- Vapor Pressure (required if checks and warnings for cavitation are desired)

The Fluid Library window also allows you to open the Change Fluid Data window which is essentially the same window as the Add New Fluid window. For easy editing, data for the fluid you wish to change is entered for you in the appropriate areas. The original curve fit data you entered is also displayed.

The Add New Fluid/Change Fluid Data windows offer cross-plotting of your custom curves against the curve fit data points. You can also import data from text files into these windows.

Related Topics

Library Manager.....	708
Library Browser.....	708
Pipe Material Library.....	714
Junction Library.....	712
Insulation Library.....	717
Fittings & Losses Library.....	717
Libraries.....	707
Network Library Overview.....	723
Library Verification.....	727
Library Locations in General Preferences.....	729
Pipe Material Library References.....	766

Pipe Material Library

Adding Custom Pipe Materials to the Pipe Materials Library

You can open the Edit Pipe Materials panel from the [Library menu](#). Using this panel, you can easily add to the user library the geometric information for any pipe material in the length units of your choice. Custom pipe materials can be selected for use in the [Pipe Properties window](#), minimizing the data entry required by the user.

Pipe materials typically have a range of nominal sizes and a specific type, class, or schedule that describes the actual dimension. AFT Fathom works with the inner diameter of the pipe, since that is the parameter that directly affects the fluid mechanics.

To add custom pipe materials to the user library, choose the entry location you desire in the list on the left (Pipe Materials, Sizes and Types), then click the New button below. For example, to add an entirely new material, click the Pipe Materials item at the top of the list then click the New Material button. To add a new size for an existing material (say Steel), click *Steel* in the list then click the New Size button. To add a new type to the Steel 1 inch size, click *1 inch* under the *Steel* material in the list then click the New Type button.

When adding a new material, you also will be asked to specify a geometry, enter a nominal size, type of material, diameter, friction data set, and friction value to define the first data. Optionally you can add in a wall thickness. From then on you can enter additional nominal sizes and types. The name descriptors you choose for the material, size, and type are at your discretion. They can contain text mixed with relevant numbers if you desire.

You can also add multiple Friction Data Sets for each pipe material. This allows you to keep, for example, different friction values that may apply to the same pipe under different conditions. These conditions could be related to the age of the pipe or the type of fluid being carried. See the Friction Data Sets section below for more information.

In addition to manual entry of the custom pipe materials, you also can import pipe material data from a file. When you click the Import From File button, a dialog box will appear explaining the necessary import file format. By choosing to import from a file, you will be able to have a separate copy of the data for later access.

Friction Data Sets

Friction Data Sets are pipe friction values for a particular pipe condition.

To create a new Friction Data Set select a Pipe Material, Size, or Schedule to associate the data set with, then click New Data Set from the Friction Data Set tab on the right (Figure 1). Modify the Friction Data Set description to provide a name, then enter the Friction Model and Value.

A Friction Data Set can be associated with the entire pipe material, or with specific sizes or types. For instance, a friction data set associated with Steel (i.e., the top level) might be called "25 year usage" and have a roughness value of 0.005 inches. This data set is then available to all sizes and types of steel. However, if "25 year usage" is associated only with the 1 inch of Steel, it will apply only to the pipe schedules that are 1 inch.

You can also associate friction data sets with a specific type (i.e., schedule). For instance, you can make a "25 year usage" and associate it only with 1 inch schedule 40 Steel pipe and assign its roughness value as 0.006 inches. Then all Steel pipe would use 0.005 inches for "25 year usage" friction while the different value for 1 inch schedule 40 of 0.006 inches would supersede the value of 0.005 inches associated with all Steel pipe.

Finally, among multiple friction data sets you can specify a default data set.

Multiple friction data sets can be created for each type of pipe material. For example, there may be separate friction data sets for Steel pipe such as: New Pipe, 10 Year Use, 20 Year Use and 30 Year Use. Typically, the older pipes will have higher friction.

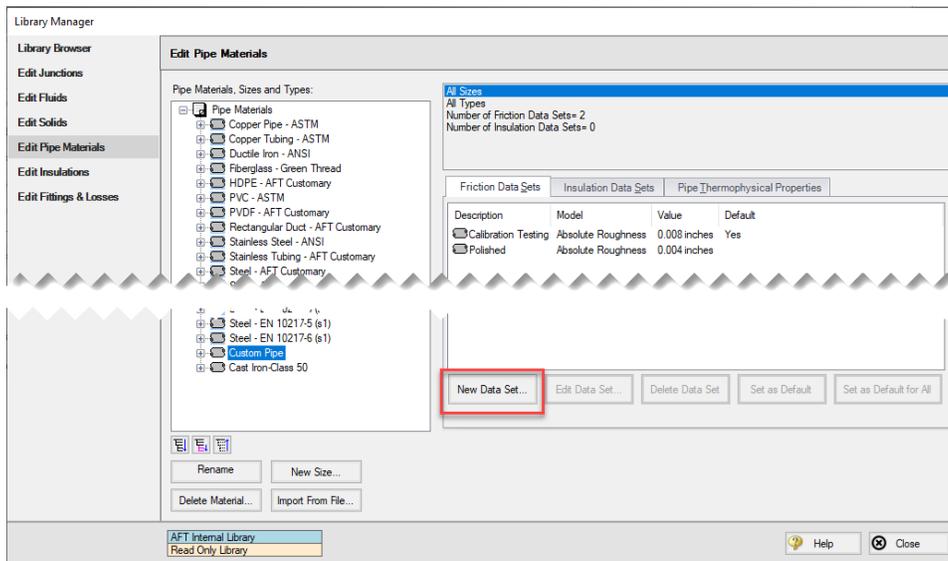


Figure 1: Friction Data Set tab on the Edit Pipe Materials panel

Insulation Data Sets

Insulation data sets are similar to friction data sets. An insulation data set can be associated with the entire pipe material, or with specific sizes or types. Insulation data sets are based on insulation data that has been specified in the [Insulation Library](#). An insulation type must be defined first in the Insulation Library before it can be used for an Insulation Data Set.

To create an insulation data set select the pipe material, schedule, or type that will be associated with the data set in the list on the left, then click the Insulation Data Set tab on the right and select the New Data Set button. Modify the Insulation Data Set description to provide a name, then choose an Insulation Library from the drop-down and define an Insulation thickness value. Once an insulation Data Set is defined it can be selected for use in the [Pipe Properties window with Heat Transfer enabled](#).

As an example, an insulation data set associated with Steel (i.e., the top level) might be based on insulation data in the Insulation Library called "Denver Heating Company K-Series". You might assign it a thickness of 1 inch and name it "Colorado Standard". Once entered, all steel pipe sizes would have a Colorado Standard insulation data set based on Denver Heating Company K-Series insulation that is one inch thick.

In practice, different thickness would likely be associated with different pipe sizes. To accomplish this, you could create insulation data sets for pipes one inch or less, call them "Colorado Standard", and make them ¼ inch thick. Pipes 1-4 inches could also have "Colorado Standard" sets, but would use ½ inch insulation, and pipes larger than 4 inch could have "Colorado Standard" and be specified as 1 inch thick. Then in the Pipe Properties window you can choose "Colorado Standard" insulation, and the appropriate thickness for the size you have chosen would be used.

Finally, among multiple insulation data sets you can specify a default data set.

Pipe Thermophysical Properties

The Edit Pipe Materials panel allows you to define thermophysical properties for the pipe. The density of the pipe can be entered as a constant, and the thermal conductivity as a function of temperature. The thermal conductivity is used in [heat transfer calculations for pipes](#).

To add thermophysical properties select a Pipe Material to associate the data set with, then click Specify Properties from the Pipe Thermophysical Properties tab on the right. The Pipe Physical Properties window will be shown where the thermal conductivity data can be entered on the Pipe Material Property Data tab, and the density can be entered on the Density Data tab. A density calculator is provided to calculate the density using the pipe weight and dimensions.

Related Topics

Library Manager.....	708
Library Browser.....	708
Fluid Library.....	713
Junction Library.....	712
Insulation Library.....	717
Fittings & Losses Library.....	717

Libraries.....	707
Network Library Overview.....	723
Library Verification.....	727
Library Locations in General Preferences.....	729
Pipe Material Library References.....	766

Insulation Library

The Edit Insulations panel is accessed from the [Library menu](#). This panel allows you to view the insulation data provided with AFT Fathom, or to add, edit or delete user defined insulation materials. Insulation library items are used for heat transfer calculations in pipes.

To add a new insulation material click Add New Insulation, then enter the data in the Insulation Physical Properties window. The insulation thermal conductivity is entered as a function of temperature, and density as a constant.

Insulation data can be applied directly to pipes, and can also be associated with the [Pipe Material Library](#) as insulation data sets.

Related Topics

Library Manager.....	708
Library Browser.....	708
Fluid Library.....	713
Pipe Material Library.....	714
Junction Library.....	712
Fittings & Losses Library.....	717
Libraries.....	707
Network Library Overview.....	723
Library Verification.....	727
Library Locations in General Preferences.....	729
Pipe Material Library References.....	766

Fittings & Losses Library

The Edit Fittings & Losses panel can be accessed from the [Library Menu](#) and allows the user to review the [Fittings & Losses](#) data provided by default in Fathom, and to add, edit or delete user defined Fitting & Losses in the AFT Default Internal Library.

Fittings & Losses can either be added by using the entry form in the window, or by importing a text file as described below. To enter a new Fitting/Loss item in the Edit Fittings & Losses panel click Create New Loss Item. Define the applicable category, type, geometry and loss model information then click Accept. The user-defined item will appear in the Fittings & Losses on the left under the relevant category/type with the source shown in parentheses as "U".

Custom fittings & losses are saved to the [local user library](#). From there the user can move them to other libraries (see [Library Manager](#)).

Import From File

Fittings & Losses can be imported from a *.CSV type file, which can be created from any spreadsheet-type software, such as Excel.

The entries in the file should be in the order as shown in Figure 1, but with no headers in the workbook. If a header row or a blank row is used, Fathom will present a warning, but give the user the option to still import the valid rows.

	A	B	C	D	E	F	G	H
1	Category	Type	Sub Type (Optional)	Geometry (Optional)	Geometry Value (Optional)	Loss Model	Loss Value	Reference/Notes (Optional)
2	Elbow/Bend	Mitre	General	Angle (degrees)	45	K (constant)	5	

Figure 1: Example of import file setup for a custom elbow/bend. Headers should not be present in the file to import.

Valid options for each of the columns are listed below, with detailed options given in Table 1.

- **Category** - Must match one of the values from Table 1. "Other" may be used if none of the specific fitting/loss categories fit.
- **Type** - This field is required, any name/value may be entered. Types already existing in the library are shown in Table 1, and may be used if desired.
- **Sub Type** - This field is optional. A sub-type may be chosen from Table 1, or any other name/-value may be entered.
- **Geometry** - This field is optional. A geometry type from Table 1 must be used if a geometry is entered. If the Geometry field is left blank, the Geometry Value will be ignored.
- **Geometry Value** - This field is only required if a Geometry type has been entered. Any positive numerical value may be entered.
- **Loss Model** - This field is required. The Loss Model must be either "K (constant)" or "K / fT (Crane)".
- **Loss Value** - This field is required. Any positive numerical value may be entered for the loss value.
- **Reference/Notes** - This field is optional. Any information may be entered here as needed.

Table 1: Valid Entries for each field. The Geometry Value and Loss Value fields will accept any numerical entry. The Reference/Notes field will accept any entry.

Category	Type	Sub Type	Geometry	Loss Model
Elbow/Bend	Standard Threaded	General	Angle (degrees) Pipe Size (nominal) None	K (constant) k / FT (Crane)
	Mitre	General		
	Smooth Flanged	r/D=1		
		r/D=1.5		
		r/D=2		
		r/D=3		
		r/D=4		
		r/D=6		
		r/D=8		
		r/D=10		
3-K (Darby)	90 deg			
	45 deg 180 deg			
Valve	Angle	Type 1	Angle (degrees) Open Percentage L/D Pipe Size (nominal) None	K (constant) k / FT (Crane)
		Type 2		
		Direct flow		
	General			
	Ball	None		
Butterfly	Plane disk - AR=0.92			
	Plane disk			

Libraries

		t/D=0.1 t/D=0.15 t/D=0.2 t/D=0.25 t/D=0.3 t/D=0.35 Type A Type B Type C		
	Cylindrical	General		
	Gate	Rect. Tube		
		Wedge/Plug		
		Wedge		
		Disk w/o recess		
		Disk with recess		
		Disk		
	Globe	Wedge - Full Open 10 deg.		
		Wedge - Full Open 20 deg.		
		Sluice		
		Disk with recess		
	Plug	90 degree wall		
		45 degree wall		
		Standard 45 deg. wall		
	Poppet	Standard 90 deg. wall		
		Standard Y Type		
		Straight-way		

Libraries

	Three-way	W/D=.035 W/D=.07 No Seat Width		
	3-K (Darby)	Through flow Branch flow		
		Angle Globe Plug Gate Ball Diaphragm		
Check Valve	Swing	90 deg. seat 45 deg. seat	Angle (degrees) None	K (constant) k / FT (Crane)
	Lift	90 deg. wall 45 deg. wall		
	Tilting disc	General		
	Stop-check	Globe 90 deg. Globe 45 deg. Globe thru flow		
	3-K (Darby)	Swing Check Lift Check		
Orifice	Sharp	None	Area Ratio	K (constant)
	Long	None	None	k / FT (Crane)
Entrance/Exit	Entrance	Sharp Reentrant Rounded Flush	R/D None	
	Exit	None		
Other	Diff Flowmtr	Orifice Plate	Area Ratio	K (constant)
		Nozzle	None	k / FT (Crane)

Libraries

		Venturi 7 deg. Venturi 3 deg. Dall Tube		
	Honeycomb cell	L/D=2 L/D=4 L/D=6 L/D=8 L/D=10 L/D=12		
	Screen	Perforated Plate Circular wire Silk thread Round wire or netting		
	Tee	Through flow (dead branch) Branch flow (dead run)		
Flange	Steel	150 lb. 300 lb.	None	N/A

Related Topics

Fittings & Losses.....	218
Equivalent Length.....	267
Library Manager.....	708
Library Browser.....	708
Fluid Library.....	713
Pipe Material Library.....	714
Junction Library.....	712
Insulation Library.....	717
Libraries.....	707
Network Library Overview.....	723
Library Verification.....	727
Library Locations in General Preferences.....	729
Pipe Material Library References.....	766

Related Examples

Plant Cooling Cost Calculation	958
--------------------------------------	-----

Network Library Overview

Custom libraries contain custom data generated by the user, including fluid properties, pipe materials, and data for junctions types such as pumps and valves.

All custom data created by the user is kept on the local computer in a file called FTH_USER11.DAT in the local user directory. This data is not shared and is always available to the local user for access or for editing.

In addition to the local user library, you can create multiple network or local custom libraries. The [Library Manager](#) window on the Library menu simplifies this process. See [Creating an Enterprise-wide Network Library System](#) for details on how to do this.

Related Topics

Benefits of Shared Libraries.....	723
Creating an Enterprise-Wide Network Library System	724
Libraries	707
Library Manager.....	708
Library Browser.....	708
Library Verification.....	727
Library Locations in General Preferences.....	729
Fluid Library.....	713
Pipe Material Library.....	714
Pipe Material Library References.....	766
Junction Library.....	712
Insulation Library.....	717
Fittings & Losses Library.....	717

Benefits of Shared Libraries

When the Fathom libraries are placed on a network, powerful data management procedures can be implemented. Specific, quality-checked data can be entered into libraries located on a local or wide area network of PC's. This data is not accessible to the end users through AFT Fathom itself, so it is guarded data.

Once the data is placed on a network, users can connect to the library in the [Library Manager](#). Users can connect to as many libraries as they want.

If the user connects to a library with these files and selects to be connected to any of these control files, the model will inherit the control formatting specified in the control files. This means that special report layouts, special graphical representations and colors, and other special controls can be decided upon by the library constructor, and those control features can be loaded into local models by end users. This is highly useful for creating customized reports for certain customers or project requirements.

By way of example, assume the [Output Control](#) content was specified as coming from the "Colorado Springs Office Library" library.

If you uncheck the Library checkbox or change certain data in the Output Control window, the library will remain connected but change to an inactive state. When the library is inactive, none of its data transfers to the model. To bring the Output Control data back into sync with the connected library, recheck the Library check box in Output Control.

This entire custom library concept lends itself naturally to the idea of specific project libraries. Each project has differing requirements that depend on the nature of the project and the end customer. Data for each project can be assembled into separate libraries, allowing faster and more accurate transmission of the controlled data and desired reporting formats to engineers.

Maintaining a network library allows commonly-used data to be controlled, which may be helpful in instances where quality control is strict.

Related Topics

Network Library Overview.....	723
Creating an Enterprise-Wide Network Library System.....	724

Creating an Enterprise-Wide Network Library System

Overview

Users of AFT Fathom on a network can access common libraries to simplify model verification, improve analysis and reporting consistency, and eliminate redundant work.

To accomplish this there is some initial setup that must be done. First, the data for components, materials, formats, etc., needs to be created and saved to various data files. Secondly, these files must be made available to the users of AFT Fathom by creating a file named DATABASE.LIB in the FATHOM folder. Thirdly, the users need to connect to the new libraries listed and select the sections to use. Each of these broad steps is discussed below. To maintain control and integrity of the data, it would be advised that one person is given the responsibility to create and maintain the network libraries. The discussion below will be given from the viewpoint of this person.

Table 1 shows the relationship between Internal Libraries and External Libraries as well as the library file listing that AFT Fathom uses to find these libraries. An External Library is any library file that is created by the user and listed in the FTH_DBUSER.LIB residing in the user's local user directory. The user can make these available and unavailable, and connect and disconnect from these libraries using [Library Manager](#). The library files themselves may be located on any local or mapped drive. These library files are also distinguished by being under the local user's complete control. External Shared Libraries, on the other hand, are common to all AFT Fathom users on the network, therefore the local user cannot remove them from the list of available libraries. However, the user can choose to connect or disconnect from these libraries using Library Manager. Finally, to complete the discussion, there are two special libraries, the Default Internal Library and the [Local User Library](#). These files are AFT Fathom Internal Libraries which cannot be disconnected or made unavailable.

Table 1: Summary of the types of libraries used for the example below, note the difference between External and External Shared libraries is just which *.LIB file their file path is listed in

Library Groups & Filenames	Library Titles	Library Locations
Fathom Internal Libraries		
FATHOM#.DAT FTH_USER#.DAT	AFT Default Internal Library AFT Fathom Local User Library	AFT Fathom # folder, where # is the version number AppData folder
External Shared Libraries (DATABASE.LIB)		
VailMaterials.dat	Vail Resort Materials	Network Location, with path listed in DATABASE.LIB
External Libraries (FTH_DBUSER.LIB)		
VailMaterials.dat	Vail Resort Materials	Network Location, with path listed in FTH_DBUSER.LIB

Creating Library Files

The first thing to do is create the files that contain the information to be shared. These files will be referred to as library files and can contain information about pipe materials, components (e.g. pumps, valves, etc.), fluids, etc. The following are several ways to create library files:

- Use the Create New Library button in the Library Manager, as described previously. Data can be taken from the current model or other libraries.
- Copy and paste sections from one file into another file using a text editor.

This discussion will focus on the first and second methods. Let's assume that the person responsible for the network library has already created a new pipe material. Also, assume that the goal is to create a network-wide library to contain this material.

The information for the material is copied from the [Local User Library](#) (Fth_User#.DAT, where # is the corresponding version number) to a new file using the [Library Browser](#) in the Library Manager. For the source library choose Local User Library and then select Pipe Materials. Then click Create New Library and enter a file name, VailMaterials.dat, for this example. Enter a description of the library - this description is how AFT Fathom users will refer to the library and should concisely describe what is in the library (for this example we'll use "Vail Resort Materials"). Right-click the pipe materials section from the Local User Library and choose Copy Content to copy the pipe materials to this new library file. The pipe material library file has now been created and is ready to be used.

Sharing Library Files Using DATABASE.LIB

Now that the library file for the Pipe Materials has been created, it needs to be made available to the users of AFT Fathom. Depending on the scope of use, library files can be placed into one of two groups. If the library is to be used locally by one user, it is put in the External Library group. On the other hand, if the library is to be common among many users and shared over a network it is put into the External Shared Library group.

Continuing the above example, the goal is to share the library file, so it will need to be an External Shared Library listed in the DATABASE.LIB file in the Fathom directory on the network. However, since Fathom does not write directly to the network files for security reasons, it is easiest to temporarily treat these files as local External Library files which will be listed in the FTH_DBUSER.LIB file located in the local user directory of the local machine.

It is recommended that External Shared Library files be copied to the network server and placed in a subfolder of the AFT Fathom folder (e.g. AFT Products\AFT Fathom\Libraries). The files may, however, be located anywhere on the network that is accessible to the users. For our example, we are going to put the pipe material library on the network in a subfolder called Libraries. It is easier to copy the file to the Libraries folder before they are made available in AFT Fathom. This will help establish the correct paths to the files as the files are made available using the Library Manager.

Once the library file is in the network location, open the [Library Manager](#) in AFT Fathom to the Library Browser. Each of the new library files we have created needs to be added to the Available Libraries list by clicking Add Existing Library and browsing to the file. Once you click OK, the description of the library you just added appears in the Available Libraries list. Once you have added the new library, click Close and the External Library listing file (FTH_DBUSER.LIB) in your local user directory is automatically updated.

Now we need to discuss how the paths should be setup. When you add a library, AFT Fathom will write the full path of the library to the External Library listing (FTH_DBUSER.LIB). That means that if you browsed to the library file using a mapped drive letter then the path will include that drive letter. However, if other users try to access the libraries but have the AFT Fathom folder mapped to a different drive letter they will not be able to locate the library files. Other options include using either relative paths or the computer name in the path (e.g. \\Network Server\Eng Apps\AFT Products\AFT Fathom\Libraries\filename.dat). You can automatically have AFT Fathom use the computer name when you add the file by choosing Network Neighborhood, then the computer and then browse to the library file. If you would rather use relative paths, you can manually edit them in the library listing file. All library files must be relative to the AFT Fathom folder where the executable is located.

When the pipe materials library was added it became an External Library listed in FTH_DBUSER.LIB. Now we need to move this listing from the local machine to the AFT Fathom folder on the network so that all users can access the library files. Once the listing file is moved to the network, it must be renamed DATABASE.LIB. Once this is done, all users can now access the library file we have created.

Connecting to the External Shared Libraries

Now that the libraries have been placed on the network and made available to all AFT Fathom users, each user must now connect to the libraries and make the sections active as needed. When a user opens the Library Manager, the External Shared Libraries are listed in the Library Browser. Make sure the checkbox is checked to make those libraries connected for use. Also ensure that the sections within the library are checked for use. A library can be connected without having any sections selected and active.

Finally, to have these connections in place every time you start AFT Fathom, click Set As Default. To apply these settings to a model made previously, open the model, go to Library Manager, click User Default and, if you have set the default, the connections and setting will now be applied to the model.

Sharing Model Formats/Preferences

Note that the Library Manager used to provide the option to include formats/preference settings such as the Output Control layouts into shared libraries. These type of files can still be shared, but are no longer shared through the Library Manager. Instead these type of format files must be loaded directly through their control windows, such as loading the Output Control format being loaded in the Output Control window.

Related Topics

Network Library Overview.....	723
Benefits of Shared Libraries.....	723

Library Verification

The Library Verification window is displayed when you open a model or new scenario which contains connected library items that have some inconsistency.

1. Library Item Not Found

The model file may have a junction which is derived from a library, but the referenced library is not currently connected in the [Library Manager](#). For example, a valve junction may be connected to a library, and derive its data from that library. However, when the model is loaded, the source library is not available. This could be for several reasons:

- The library is no longer available because it was removed either through the Library Manager or because your libraries have been changed on the network.
- The model is being run on a different PC than where it was created, and the original libraries are no longer available. This could be, for example, because you have moved the model onto a laptop computer to take it on travel. Or it could happen because the model was developed by another company which had developed libraries of its own but which are not available to you.

The valve junction in the model does in fact have all of the required data needed to run the model; however, when it is not found in the library, you will be offered several options:

- **Add To Library** - The junction(s) will be automatically added to the [Local User Library](#), from which the contents can be moved to a different library if desired. If the model file is saved, the junction(s) will now associate the library connection with the Local User Library, and not the original library which is currently unavailable.
- **Disconnect** - The model will disconnect the junction from the library.

- **Keep Name** - The junction will keep its original name, remember what library to which it was originally connected, and connect to that library again when it is made available. If, however, you try to open the junction Properties window, it will become disconnected from the library.

Whatever choice you make, the model will retain all of its engineering data and yield the same results as it did originally.

2. Item in Model is Different from Library

The model file may have a junction which is derived from a library, and the referenced library is connected, but for some reason the data is not the same. For example, you may have a valve junction with a Cv vs Open % curve for the Rocky Mountain Valve Company's Model 100A in your model, but it does not match the curve data for this item in your library. This could be for several reasons:

- When the model was built, the valve was connected to the Rocky Mountain Valve Company (i.e., RMVC) Model 100A library item. But since that time, the RMVC library has changed. Perhaps there was an input error in the original data, or better data has since become available, and the library was updated. If this were the case, it would be best to update all your valves to the library values.
- The library may have been changed by an unauthorized individual, and while it was originally accurate, now the data is no longer valid. In this case, it would be best to **not update** the valve in your model, and to inquire into the cause of the inaccurate data within your organization.

Upon loading the model you will be offered several options:

- **Update Junction** - This will actually change the model's input data, such that it is replaced with the data currently in the connected library, and it might yield different results for the model.
- **Update the library (if it is local) to match the model's items** - This will change the data in the library, but will leave the model's input data as it was. The model will yield the same results as it did previously, but the library item will be updated.
- **Disconnect** - This will disconnect the junction from the library, leave the model as it was, and it will yield the same results.

Related Topics

Libraries.....	707
Library Manager.....	708
Library Browser.....	708
Network Library Overview.....	723
Library Locations in General Preferences.....	729
Fluid Library.....	713
Pipe Material Library.....	714
Pipe Material Library References.....	766
Junction Library.....	712
Insulation Library.....	717

Library Locations in General Preferences

When performing an engineering analysis with or even without cost estimating, there may be many engineering and [cost libraries](#) that underlie the model. These libraries need to be connected in the [Library Manager](#) in order to be accessible by the model. However, to know what libraries are actually needed, one needs to first load the model. This creates a dilemma.

To help alleviate the dilemma, users can create "Library Search Locations" in the [User Options](#) window. Figure 1 shows an example. Once specified, you can set these as your default locations by clicking the Set As Default button. When AFT Fathom tries to open a model that requires certain libraries, and those libraries have not been connected already, AFT Fathom looks for them in the locations specified in the User Options. In addition, AFT Fathom looks in the model's folder. If AFT Fathom finds the libraries in one of these locations, it automatically adds it to the list of connected libraries in the Library Manager. If it cannot find a library, it informs you once the model is loaded.

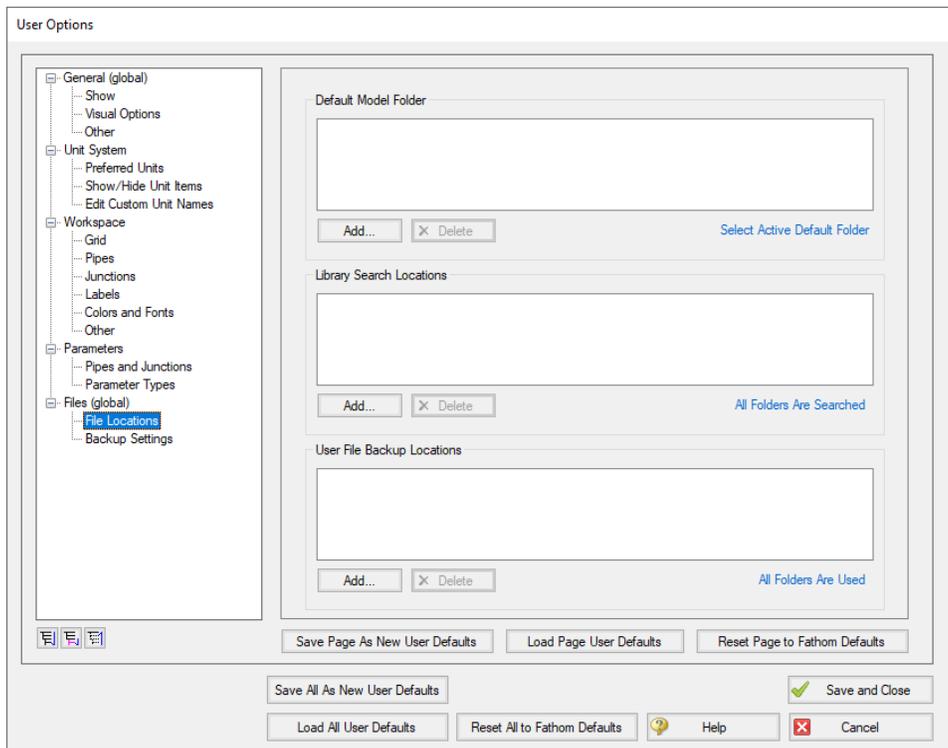


Figure 1: Library Search Locations can be defined in the User Options window

Related Topics

Libraries..... 707

Library Manager..... 708

Library Browser..... 708

Network Library Overview..... 723

Libraries

Library Verification	727
Fluid Library.....	713
Pipe Material Library.....	714
Pipe Material Library References.....	766
Junction Library.....	712
Insulation Library.....	717
Fittings & Losses Library.....	717

Theory and References

This section focuses on the fundamental theories that are used in AFT Fathom's solution method. Additionally, this chapter gives information on the source material for the theories used. Topics include discussions on the theories that comprise the network solution, detailed information on theory, and information on references used in the development of AFT Fathom.

Content

Nomenclature	731
Network Solution Methodology	733
References	741
Chempak Technical Information - Detailed Discussion	743
Role of Pressure Junctions - Detailed Discussion	755
Pipe Material Library References	766

Nomenclature

A	cross-sectional flow area of a pipe
c_p	specific heat, constant pressure
C_D	discharge coefficient
C_V	valve coefficient
CHW	Hazen-Williams factor
D	diameter of a pipe
f	friction factor
f_T	turbulent friction factor (used by Crane, 1988)
f_L	laminar friction factor
f_{Tr}	transition friction factor
F	error value in mass balance
Fr	Froude number
g	gravitational constant
h	convection coefficient

h	enthalpy
H	head
HGL	hydraulic gradeline (same as head)
J	Jacobian matrix
k	thermal conductivity
K	loss factor
K	constant for Power Law fluid
$K_{sprinkler}$	fire sprinkler loss factor
L	length of a pipe
\dot{m}	mass flow rate
n	general constant
n	constant for Power Law fluid
Nu	Nusselt Number
P	pressure, static
P_h	heated perimeter
P_o	pressure, stagnation
P_w	wetted perimeter
q	heat rate to a pipe
Q	volumetric flow rate
q''	heat flux
r	radius
r	relaxation
R, R'	resistance
R_c	coefficient of rigidity
Re	Reynolds Number
R_h	Hydraulic radius (equal to radius divided by 2)
s	pump speed
S	Submergence
S	Hydraulic gradeline gradient
S_y	yield stress

T	temperature
U	overall heat transfer coefficient
V	velocity
x	distance along pipe centerline
z	elevation
a, m, q	angle
b	diameter ratio
e	roughness
n	kinematic viscosity
ρ	density
μ	dynamic viscosity

Subscripts

i	junction at which solution is sought
j	junctions with pipes connecting to junction i
o	stagnation
∞	infinity, far away, ambient

Network Solution Methodology

AFT Fathom makes use of standard matrix solution techniques ([Jeppson 1976](#)). The method is known as the H-Equation method, where H , the piezometric head, is solved for at each junction by forcing continuity of flow through each connecting pipe. Simultaneously, the head loss across each pipe is updated based on the flow balance information. The flow rate and head are solved in an inner-outer loop algorithm, where the flow is guessed, the head loss is calculated consistent with that guess, and the flow is updated according to the new pressure drop information. The Newton-Raphson method is employed to refine each successive solution, resulting in a sparse square matrix that is solved during each solution pass.

The concepts of pressure and hydraulic gradeline (HGL , also called piezometric head) are related but use different frameworks for considering pipe system behavior. The HGL includes both the static and elevational effects of pressure. The relationship between the two is given by Equation 1:

$$HGL = \frac{P}{\rho g} + z \quad (1)$$

where z is the elevation and P is the static pressure. Each concept offers certain advantages.

The solution technique makes use of the continuity and one-dimensional momentum equations. AFT Fathom uses pressure and mass flow rate as the unknown parameters because they are more fundamental in nature than the head and volumetric flow rate used by Jeppson and thus extend more naturally to mechanical and chemical systems.

In the following discussion, subscripts denote values at junctions. Thus, P_i represents the static pressure at junction i . Double subscripts denote values along pipes connecting two junctions. Thus, \dot{m}_{ij} represents the mass flow rate in the pipe connecting junctions i and j .

Application of the law of mass conservation to each junction yields:

$$\sum_{j=1}^n \dot{m}_{ij} = 0 \quad (2)$$

where n is the number of pipes connected to junction i . Equation 2 states that the net mass flow rate into each junction must sum to zero.

The basic equation for pipe pressure drop due to friction can be expressed with the Darcy-Weisbach equation:

$$\Delta P_f = f \frac{L}{D} \left(\frac{1}{2} \rho V^2 \right) \quad (3)$$

where ΔP_f is the [frictional pressure loss](#). The total pressure change between junctions is given by the momentum equation in the form of the Bernoulli equation:

$$P_1 + \frac{1}{2} \rho V_1^2 + \rho g z_1 = P_2 + \frac{1}{2} \rho V_2^2 + \rho g z_2 + \Delta P_f \quad (4)$$

The static (normal), P , and stagnation (pressure if all kinetic energy were converted isentropically into pressure), P_o , pressures are related as follows:

$$P_o = P + \frac{1}{2} \rho V^2 \quad (5)$$

Substituting Equation 5 into 4

$$P_{o,1} + \rho g z_1 = P_{o,2} + \rho g z_2 + \Delta P_f$$

Solving for the frictional pressure drop for a constant area pipes yields:

$$P_{o,i} - P_{o,j} + \rho g (z_i - z_j) = \Delta P_f \quad (6)$$

where i and j denote upstream and downstream junction values.

The definition of mass flow rate is:

$$\dot{m} = \rho A V \quad (7)$$

Combining Equations 3 and 6 and substituting for velocity, V , using Equation 7 gives the mass flow for each pipe:

$$\left[\frac{P_{o,i} - P_{o,j} + \rho g (z_i - z_j)}{R_{ij}'} \right]^{\frac{1}{2}} = \dot{m}_{ij} \quad (8)$$

where R_{ij}' is the effective flow resistance in the pipe and the subscript ij refers to the pipe connecting junctions i and j .

$$R_{ij}' = \left(\frac{f_{ij} L_{ij}}{D_{ij}} + K_{ij} \right) \frac{1}{2 \rho A_{ij}^2} \quad (9)$$

Note that R' is *not the same* as R , which is the resistance used in the Output Control window and is expressed by the following equation:

$$\Delta H = R Q^2$$

Substituting Equation 8 into Equation 2 results in the equation to be applied to each junction i :

$$\sum_{j=1}^n \left[\frac{P_{o,i} - P_{o,j} + \rho g (z_i - z_j)}{R_{ij}'} \right]^{\frac{1}{2}} = 0 \quad (10)$$

where n is the number of pipes connected to junction i .

To be completely general, Equation 10 should be written for junction i :

$$\sum_{j=1}^n \left[\frac{P_{o,i} - P_{o,j} + \rho g (z_i - z_j)}{R_{ij}'} \right]^{\frac{1}{2}} = \dot{m}_{i,Applied} \quad (11)$$

to allow for application of boundary condition flow rates to a boundary junction node.

Equation 11 as applied to each junction in the network represents the system of equations that need to be solved to determine the stagnation pressure at each junction. To solve this system, the Newton-Raphson method is employed. In the Newton-Raphson method, new values for each unknown are calculated based on the previous value and a correction that uses the first derivative of the function.

In this instance the function that Newton-Raphson is applied to would be of the form:

$$F_i = \sum_{j=1}^n \left[\frac{P_{o,i} - P_{o,j} + \rho g (z_i - z_j)}{R_{ij}'} \right]^{\frac{1}{2}} - \dot{m}_{i,Applied} \quad (12)$$

The method involves finding all the junction stagnation pressures, $P_{o,j}$, that cause all of the F_j to go to zero, thus satisfying Equation 11 at all junctions.

When applied to a system of equations, the Jacobian matrix contains all the required derivative information to employ the Newton-Raphson technique. The Jacobian, J_F , is given by:

$$J_F = \begin{bmatrix} \frac{\partial F_1}{\partial P_{o,1}} & \frac{\partial F_1}{\partial P_{o,2}} & \dots & \frac{\partial F_1}{\partial P_{o,n}} \\ \frac{\partial F_2}{\partial P_{o,1}} & \frac{\partial F_2}{\partial P_{o,2}} & \dots & \frac{\partial F_2}{\partial P_{o,n}} \\ \vdots & \vdots & \ddots & \vdots \\ \frac{\partial F_n}{\partial P_{o,1}} & \frac{\partial F_n}{\partial P_{o,2}} & \dots & \frac{\partial F_n}{\partial P_{o,n}} \end{bmatrix}$$

The column matrix \vec{P}_o contains the stagnation pressure at each junction, and column matrix \vec{F} contains the F values (Equation 12) at each junction. The updated solutions for \vec{P}_o are obtained from the following Newton-Raphson equation:

$$\vec{P}_{o,new} = \vec{P}_{o,old} - J_f^{-1} \vec{F} \quad (13)$$

Traditional implementations of the Newton-Raphson method require initial guesses at the solution before the Solver can begin. This often results in the situation where you have to know the solution ahead of time in order to solve for it. To avoid placing this burden on the user, AFT Fathom uses a proprietary method for generating first guesses. This method is usually sufficient to get the solution going in the right direction, especially for systems that don't have highly non-linear features.

However, there are times when AFT Fathom's first guess does not lead to a converged solution. In these cases, AFT Fathom provides you with the option of specifying your own [initial guesses](#) of flow and pressure.

Related Topics

Continuity Equation at a Branch.....	737
Energy Equation at a Branch.....	737
Network Implementation.....	737
Verifying Network Solutions.....	740
Nomenclature.....	731
Head loss, head rise, hydraulic gradeline and energy gradeline output does not have meaning for zero g systems.....	786

Continuity Equation at a Branch

By applying the continuity equation at branch i , we loop over all connecting pipes and get:

$$\sum_{j=1}^n \dot{m}_{ij} = 0$$

where \dot{m}_{ij} is the flow rate in the pipe connecting junctions i and j .

Related Topics

Network Solution Methodology.....	733
Energy Equation at a Branch.....	737
Network Implementation.....	737
Verifying Network Solutions.....	740
Nomenclature.....	731

Energy Equation at a Branch

By applying the energy equation at a branch i , we loop over all connecting pipes and get

$$\sum_{j=1}^n \dot{m}_{ij} h_{ij} = 0$$

where h_{ij} is the enthalpy at the connection point of pipe and junction.

Related Topics

Network Solution Methodology.....	733
Continuity Equation at a Branch.....	737
Network Implementation.....	737
Verifying Network Solutions.....	740
Nomenclature.....	731

Network Implementation

By applying the [Continuity Equation at a Branch](#) about junction i and branches j , we get

$$\sum_{j=1}^n \dot{m}_{ij} = 0 \quad (1)$$

By applying the [Energy Equation at a Branch](#) about junction i and branches j , we get

$$\sum_{j=1}^n \dot{m}_{ij} h_{ij} = 0 \quad (2)$$

To obtain a valid solution in a network, the stagnation pressure at each branch point will be equal and the mass flow and energy flow will sum to zero at the branch. Although the stagnation pressures at the pipe endpoints will equal that at the connecting junction, this is not true for enthalpy (and temperature). The enthalpy at the end of a pipe will be that enthalpy that results from the [heat transfer in the pipe](#). Once the fluid flows into the junction, the enthalpy will change to the mixture enthalpy, or the average enthalpy of all inflowing pipes.

If only one pipe flows into the branch, the branch enthalpy will equal the enthalpy coming out of the pipe. But if more than one pipe flows into the branch, the enthalpy will reach an averaged condition. It is this averaged enthalpy that results from mixing of two or more streams at different enthalpies. This enthalpy is supplied to all outflowing pipes. The preceding discussion is depicted in Figure 1 and follows equation 3.

$$h_{mix} = \frac{\sum_{j=1}^n \dot{m}_j^+ h_j}{\sum_{j=1}^n \dot{m}_j^+} \quad (3)$$

where the + superscript indicates the summation applies only to positive inflows into the junction.

To provide converged solutions in the shortest time possible, matrix solution techniques are required. Matrix techniques allow all pipes to be solved together, which offers speed and the best convergence characteristics.

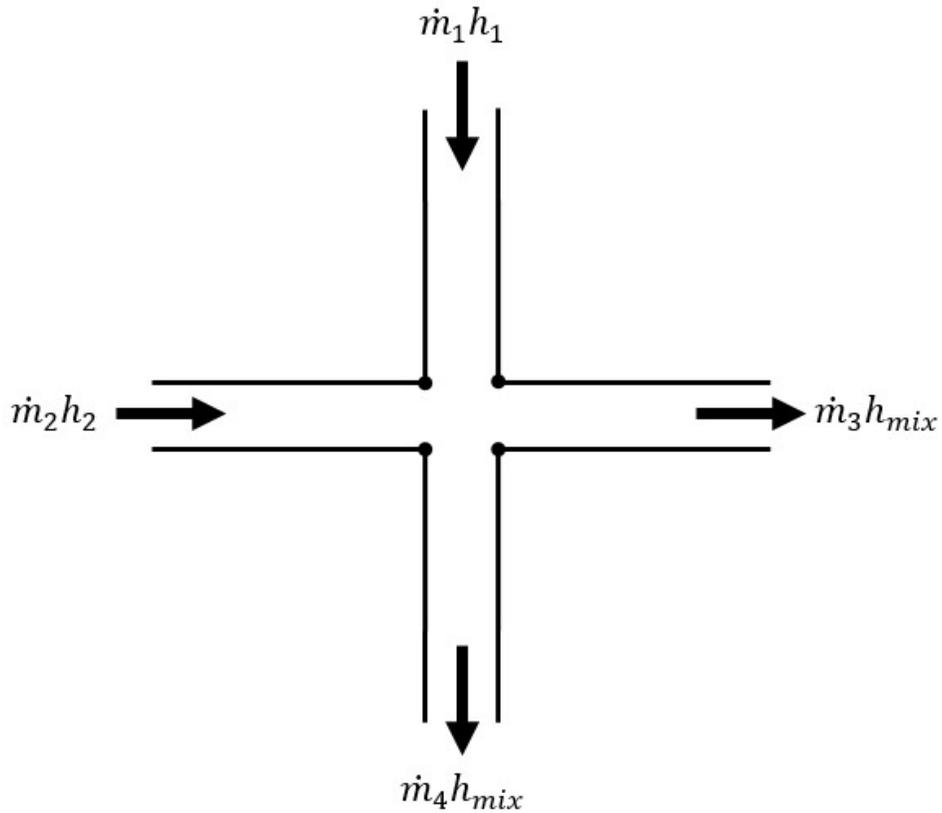


Figure 1: Flow weighted mixing and energy balance

Energy Balances with AFT Standard fluids

The AFT Standard fluids do not have enthalpy data, so Fathom uses the Reference Enthalpy Method, described in the following equation, to calculate enthalpy:

$$h = h_{ref} + C_{p,avg}(T - T_{ref})$$

Therefore Equation 3 can be expressed as the following:

$$h_{mix} = \frac{\sum_{j=1}^n m_j^+ [h_{ref} + c_{p,avg,j}(T_j - T_{ref})]}{\sum_{j=1}^n m_j^+} \quad (4)$$

The reference enthalpy and temperature is entered in the fluid library, and should be for a state that is as near as possible to the operating conditions of your model. As seen from the equation, the pressure dependence of enthalpy is ignored. In many cases, this assumption is acceptable.

Chempak and NIST REFPROP fluids have enthalpy data and Equation 3 can be used directly.

Related Topics

Network Solution Methodology.....	733
Continuity Equation at a Branch.....	737
Energy Equation at a Branch.....	737
Verifying Network Solutions.....	740
Nomenclature.....	731

Verifying Network Solutions

As discussed previously, AFT Fathom uses a Newton-Raphson matrix method to obtain a system level mass balance and energy. When the Solver (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system) is finished, the [Continuity Equation](#) and [Energy Equation](#) are satisfied at all branching sections. Once solved, it is a simple matter to go back and sum up all mass flows and energy at each branch to verify that these equations are satisfied. In fact, AFT Fathom does just that.

After the Solver converges, AFT Fathom loops over each junction and adds up the inflow and outflow of mass and energy and compares the sum to the [solution tolerances](#). If it appears that any junction is out of tolerance, a warning is given in the output.

In the Solution Balance Summary table AFT Fathom provides additional information on the balance at each junction. This table can be displayed by selecting the option in the [Output Control window](#) General tab. Figure 1 shows an example of the Solution Balance Summary table.

Jct	Connected Pipes	Mass Flow (lbm/sec)
100 (Basin 1)	100 (Circ Pump 1A Suction)	-1,319
	102 (Circ Pump 1B Suction)	-1,319
	TOTAL	-2,638
110 (Pump 1A)	100 (Circ Pump 1A Suction)	1,319
	101 (Circ Pump 1A Disch)	-1,319
	TOTAL	0.000
111 (Pump 1B)	102 (Circ Pump 1B Suction)	1,319
	103 (Circ Pump 1B Disch)	-1,319
	TOTAL	0.000
112 (Branch)	101 (Circ Pump 1A Disch)	1,319
	103 (Circ Pump 1B Disch)	1,319
	104 (Supply 1)	-2,639
	TOTAL	-0.1992
113 (Branch)	104 (Supply 1)	2,639
	105 (Cond 1 In)	-2,500
	107 (LO Clr 1 In)	-138.8
	301 (Crossover B Supply)	0.000
	TOTAL	0.08247

Figure 1: Solution Balance Summary table shown in the Output window offers a balance report of all junctions in the window

Related Topics

Network Solution Methodology.....	733
Continuity Equation at a Branch.....	737
Energy Equation at a Branch.....	737
Network Implementation.....	737
Nomenclature.....	731

References

1. *ANSI/API 526 7th Edition - Flanged Steel Pressure-relief Valves*, 2017, published by American Petroleum Institute, 1220 L Street, NW, Washington DC, USA.
2. *ANSI/API 610 11th Edition - Centrifugal Pumps for Petroleum, Petrochemical and Natural Gas Industries*, 2010, published by American Petroleum Institute, 1220 L Street, NW, Washington DC, USA
3. *ANSI/HI Standard 6.1-6.5-2015*, American National Standard for Reciprocating Power Pumps for Nomenclature, Definitions, Application, and Operation, 2015, published by American National Standards Institute, 6 Campus Drive, Parsippany, NJ, USA.
4. *ANSI/HI Standard 9.6.3-2017*, Rotodynamic Pumps - Guidelines for Operating Regions, 2017, published by Hydraulic Institute, 6 Campus Drive, Parsippany, New Jersey, USA
5. *ANSI/HI Standard 9.6.7-2021*, Rotodynamic Pumps - Guideline for Effects of Liquid Viscosity on Performance, 2021, published by Hydraulic Institute, 300 Interpace Parkway, Building A, 3rd Floor, Parsippany, New Jersey, USA.
6. *ANSI/HI Standard 9.8-1998*, American National Standard for Pump Intake Design, 1998, published by Hydraulic Institute, 6 Campus Drive, Parsippany, New Jersey, USA

7. *ANSI/HI Standard 12.1-12.6-2016*, Rotodynamic (Centrifugal) Slurry Pumps, 2016, published by Hydraulic Institute, 6 Campus Drive, Parsippany, New Jersey, USA
8. ASME Press, *ASME International Steam Tables for Industrial Use* (CRTD-Vol. 58), published by the American Society of Mechanical Engineers, Three Park Avenue, New York, NY 10016.
9. Bloch, H. P., & Geitner, F. K. (1994). *An Introduction to Machinery Reliability Assessment*. Houston, Texas: Gulf Publishing Co.
10. Brock, Pat D., *Fire Protection Hydraulics and Water Supply Analysis*, Third Edition, Fire Protection Publications Oklahoma State University, Copyright 2012 by the Board of Regents, Oklahoma State University.
11. Crane Co., *Flow of Fluids Through Valves, Fittings, and Pipe*, Technical Paper No. 410, Crane Co., Joliet, IL, 1988.
12. Darby, R., *Chemical Engineering Fluid Mechanics*, 2nd edition, Marcel Dekker, New York, NY, 2001.
13. Dezhong, L., *Flow Status Identification and Hydraulic Calculation of Tailings Slurry Pipeline*, China Minmetals, MCC China ENFI, pp 71-83, No Date.
14. Durand, R., & Condolios, E., "Etude experimentale du refoulement des materiaux en conduites en particulier des produits de dragage et des schlamms". *Deuxiemes Journees de l'Hydraulique*, 27-55, 1952.
15. Eckert, E.R.G., and Drake, R., *Analysis of Heat and Mass Transfer*, McGraw-Hill, New York, NY, 1972.
16. Garcia, E. and Steffe, J., "Comparison of Friction Factor Equations for Non-Newtonian Fluids in Pipe Flow," *Journal of Food Process Engineering*, Vol. 9, pp. 93-120, April 1986. [Online]. Available: <https://doi.org/10.1111/j.1745-4530.1986.tb00120.x>
17. Hanks, R.W. and Dadia, B.H., Theoretical analysis of the turbulent flow of non-newtonian slurries in pipes, *AIChE J.*, 17: 554-557, 1971. [Online]. Available: <https://doi.org/10.1002/aic.690170314>.
18. Idelchik, I. E., *Handbook of Hydraulic Resistance*, 4th edition, Begell House, Redding, CT, 2007.
19. Ingersoll-Dresser Pumps, *Cameron Hydraulic Data*, Liberty Corner, NJ, 1995.
20. Ito, H., "Friction factors for pipe flow," *Journal of Basic Engineering*, Vol. 81, pp. 123-126, 1959.
21. Jeppson, R. W., *Analysis of Flow in Pipe Networks*, Ann Arbor Science Publishers, Ann Arbor, MI, 1976.
22. Karassik I., Krutzsch W., Fraser W. and Messina J., *Pump Handbook*, 2nd edition, McGraw-Hill, New York, NY, 1986.
23. Miller, D. S., *Internal Flow Systems*, 2nd edition, Gulf Publishing Company, Houston, TX, 1990.
24. Mullineux, G., "Non-linear least squares fitting of coefficients in the Herschel-Bulkley model," *Applied Mathematics Modeling*, Vol. 32, Issue 12, pp. 2538-2551, Dec. 2008. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S0307904X07002235>
25. Nayyar, M. L., *Piping Handbook*, 7th edition, McGraw-Hill, New York, NY, 2000.
26. NFPA 13, *Installation of Sprinkler Systems*, 1999 Edition, page 13-139.
27. NFPA 15, *Standard for Water Spray Fixed Systems for Fire Protection*, 2012 Edition.
28. *Pipe Line Rules of Thumb Handbook*, Gulf Publishing, Houston, TX.
29. TAPPI, *Generalized method for determining the pipe friction loss of flowing pulp suspensions*, TAPPI, TIS 0410-14, Atlanta, GA, 1988.
30. Val-Matic Valve and Manufacturing Corp., *Surge Control in Pumping Systems*, Val-Matic Valve and Manufacturing Corp., Elmhurst, IL, 2009.

31. Visintainer, Robert. "GIW 4-Component Model Summary 2022-09-03." GIW Industries Inc., 2022. [Online]. Available: <https://www.ksb.com/en-us/services/service-for-slurry-pumps/hydraulic-lab>
32. Walker, C. I., *The effect of settling slurries on pump performance*, undated, Warman International.
33. Wasp, E. J., Kenny, J. P., & Gandhi, R. L., *Solid liquid flow slurry pipeline transportation*, Transactions Technical Publications, 1977.
34. Wilson, K.C., G.R. Addie, A. Sellgren and R. Clift, *Slurry Transport Using Centrifugal Pumps*, 3rd Ed., published by Springer, 2006.
35. Wilson, K.C., "Effect of Solids Concentration on Deposit Velocity", *Journal of Pipelines*, 5, 1986, pp. 251-257.
36. D. Wood, T.W Walters, "Operational problems in pumping non-settling slurries resolved using an improved laminar pipe fitting loss model" in *28th Intern. Pump Users Symp.*, Houston Texas, Turbomachinery Laboratory, Texas A&M University, 2012
37. Xiangjun, Fei, *Hydraulics of Slurry and Granular Material Transportation*, Tsinghua University Chuxin Society, 1994.
38. Zipparro, V. J., and H. Hasen, editors, *Davis' Handbook of Applied Hydraulics*, Fourth Edition, McGraw-Hill. New York, NY, 1993.

Chempak Technical Information - Detailed Discussion

© Copyright 1998 Madison Technical Software Inc. Used with permission.

Note: The information in this Appendix was taken from Section 8 of the Chempak Operating & Reference Manual, Version 4, Windows Edition, Issue: January 1998. Section numbering was left unchanged, and some sections that were judged not relevant were not included so the numbering is not sequential. Notation and references in this Appendix apply only to this Appendix, and are referenced at the end.

8.1 General

This section sets out the data sources, correlations and estimation methods used in the CHEMPAK database. In putting together the database, the methods and sources were selected in the following order of preference.

- Published experimental data
- Published correlations based on experimental data
- Specific category correlations
- General estimation methods

Madison Technical Software has followed the general recommendations in Reid and in Danner and Daubert as far as selection of specific category correlations and general estimation methods are concerned. In selecting specific compound data, a combination of sources has been used wherever possible. Important sources of specific compound data used by CHEMPAK are:

- Reid et al
- Perry et al
- J Chem Eng Data
- Daubert & Danner

- ESDU publications
- API Technical Data Book - Petroleum Refining
- International Critical Tables
- CRC Handbook
- Vargaftik

In many cases, the compound property values are a combination of published data, published correlations and general estimation methods. Several properties in certain compound categories have been estimated or adjusted by Madison Technical Software. It has been our policy to adopt and maintain a critical approach to available data sources and correlation methods.

The following sections set out details of the correlations and estimation methods used. In certain cases, the user is directed to the original references, particularly where the method is complex.

Data sources for Aqueous Solutions/Heat Transfer Liquids are published experimental data and correlations based on experimental data.

8.2 Physical Constants

8.2.1 Critical Temperature

The great majority of values are believed to be experimental. Where values had to be estimated, the Joback method was used.

8.2.2 Critical Pressure

Most of the values are experimental. In cases where experimental data were not available, the critical pressure was derived from the Joback method.

8.2.3 Critical Volume

A majority of the values are experimental. A great majority of the remaining compounds for which experimental values were not available had accurate boiling-point volumes available from which critical volume estimates were derived using the Tyn and Calus correlation. For a few substances, estimates of the critical volume were derived from the Joback method.

8.2.4 Normal Boiling Points

All values are believed to be experimental. In some cases, the values were slightly adjusted for vapor pressure.

8.2.5 Freezing Points

Where possible, quoted freezing points are experimental. No accurate method of estimation of compound freezing point is available. In the absence of experimental data, a rough estimate was derived from the Joback method.

8.2.6 Acentric Factors

The acentric factor is defined as

$$w = -\log_{10}(P_{vr} \text{ at } T_r = 0.7) - 1$$

In all cases the acentric factor was derived from the vapor pressure correlation (see section 8.8)

8.2.7 Joback Group Contribution Method

The Joback method is used to derive values of T_c , P_c , V_c and T_f where no experimental data or other predictive method was available.

$$T_c = T_b / (0.584 + 0.965 \sum(D_t) - \sum(D_t)^2)$$

$$P_c = 1 / (0.113 + 0.0032 n_a - \sum(D_p))^2$$

$$V_c = 17.5 + \sum(D_v)$$

$$T_f = 122 + \sum(D_f)$$

where n_a is the number of atoms in the molecule and the D contributions are given by Joback and by Reid et al (1987). Error magnitudes for the Joback method are as follows:

- Critical Temperature: average error about 1%
- Critical Pressure: average error about 5%
- Critical Volume: average error about 2%
- Freezing Point: average error about 11%

8.2.8 Tyn & Calus Relation

Tyn & Calus showed a close (< 3% error) relation between molar volume at normal boiling point and the critical molar volume of the form,

$$V_b = a V_{cn}$$

$$a = 0.285$$

$$n = 1.048$$

8.3 Liquid Specific Volume

Liquid specific volume rises slowly and approximately linearly with rise in temperature to about $T_r = 0.8$. At higher temperatures, the values rise more rapidly to the critical point.

Experimental data or correlations derived from experimental data are available for most compounds.

8.3.1 Hankinson-Brost-Thompson Equation

The saturated specific volume is given by,

$$V_s/V^* = V_r(O)(1 - wsrkV_r(1))$$

$$V_r(O) = \sum\{a_n(1 - T_r^n/3)\} \quad 0.25 < T_r < 0.95$$

$$V_r(1) = \sum\{b_n T_r^n / (T_r - 1.00001)\} \quad 0.25 < T_r < 1.0$$

$$a_0 = 1$$

$$a_1 = -1.52816$$

$$a_2 = 1.43907$$

$$a_3 = -0.81446$$

$$a_4 = 0.190454$$

$$b_0 = -0.296123$$

$$b_1 = 0.386914$$

$$b_2 = -0.0427258$$

$$b_3 = -.0480645$$

V^* , w_{srk} and T_c are tabulated property constants. The user is referred to Hankinson, Thompson and to Reid et al (1987). Errors are typically about 1% with most being less than 2%.

8.3.2 Rackett Equation

If a reference volume (V_{ref} at T_{ref}) is available then

$$Z_{ra} = (P_c V_{ref} / RT_c)^n$$

$$n = 1 / (1 + (1 - T_{ref} / T_c)^m)$$

$$m = 2/7 \text{ or other empirical constant}$$

The saturated specific volume is given by,

$$V_s = V_{ref} Z_{ra}$$

with

$$x = -(1 - T_{ref} / T_c)^m + (1 - T / T_c)^m$$

In most cases, an experimental value of reference density was available. Where such a value was not available, values were derived from the group contribution method of Le Bas or derived from the critical volume using the Tyn & Calus relation. Tests by Madison Technical Software on over 80 liquids showed that these two methods were significantly more accurate than the Spencer and Danner method for Z_{ra} . The reader is referred to Reid et al for further details on these methods. With one or more experimental points, the Rackett equation gives errors of about 1% with most values less than 3%. If the reference volumes are estimated, typical errors are 3%.

8.3.4 The Effect of Pressure on Liquid Specific Volume

The effect of pressure on liquid specific volume is calculated when

$$P > P_s + 0.1 P_c$$

The correction is derived from the equation of state as follows,

$$V_L = V_{Ls} - V_{Ls,es} + V_{L,es}$$

$$V_L = \text{specific volume at } T \text{ and } P$$

$$V_{Ls} = V_L \text{ at } T \text{ and } P_s \text{ from methods of this section}$$

$$V_{Ls,es} = V_L \text{ at } T \text{ and } P_s \text{ from the equation of state}$$

$$V_{L,es} = V_L \text{ at } T \text{ and } P \text{ from the equation of state}$$

Liquid specific heat can in principle be derived from the equation of state but in practise, direct analytical or group contributions are preferred where experimental data are not available.

8.4 Liquid Specific Heat

8.4.1 Rowlinson-Bondi Method

$$(C_{pL} - C_{po})/R = 1.45 + 0.45/X + 0.25w(17.11 + 25.2X0.333/Tr + 1.742/X)$$

$$X = 1 - Tr$$

$$w = \text{acentric factor}$$

This method is generally applicable to the range from T_f to values approaching T_c . Note that C_{pL} approaches infinity as T approaches T_c .

Errors are generally less than 5% except in the case of hydrogen-bonding polar compounds (e.g. alcohols) at low temperatures. For these compounds, the Missenard group contribution method is preferred.

8.4.2 Missenard Method

The Missenard group contribution method yields values of coefficients in

$$C_{pL} = a + bT + cT^2$$

The accuracy is usually better than 5%. The method cannot deal with double bonds and is not applicable for $T_r > 0.75$

$$a = \text{Sum}\{a_n\}$$

$$b = \text{Sum}\{b_n\}$$

$$c = \text{Sum}\{c_n\}$$

The group contributions are available in Missenard. See also Reid et al.

8.4.3 The Effect of Pressure on Liquid Specific Heat

As noted above, the equations of state can be employed to estimate liquid specific heat, but the methods presented in 8.4.1 and 8.4.2 are more reliable. The equations of state however can be used to estimate the effect of pressure on liquid specific heat.

$C_{pLs} = C_{pL}$ at T_s and P_s determined by the methods of this section.

C_{po} = ideal gas specific heat at T_s

$C_{pL} = C_{pL}$ at T_s and $P > P_s$

The equations of state give estimates of

$D_s = (C_{pLs} - C_{po})_{es}$ at T_s and P_s

$D = (C_{pL} - C_{po})_{es}$ at T_s and P

The corrected value of the liquid specific heat is

$$C_{pL} = C_{pLs} + D - D_s$$

The correction is not applied when T is close to T_c

8.5 Liquid Viscosity

Liquid viscosity typically varies in magnitude by a factor of 100 or more between the freezing and critical temperatures. No generalized method is available to estimate or represent liquid viscosity adequately over the entire temperature range. Corresponding states methods are applicable above $T_r = 0.76$. From the freezing point to the boiling point, the influence of structure is strong.

8.5.1 Method of Van Velzen

The method of Van Velzen et al is a group contribution method of some complexity and range of applicability. It is the most frequently used group contribution method. The accuracy of the estimation averages about 10% and most estimates are better than 20%. Some of the limitations of the method are:

- Larger errors found with the first members of a homologous series
- Only normal and iso substitutions on alkyl chains can be treated
- Heterocyclic compounds cannot be treated
- Application only in the range T_f to T_b

The method is complex and the reader is directed to the original references for full details.

8.5.2 Method of Morris

The method of Morris is a group contribution method. This method is useful as a comparison and substitute for the Van Velzen method in cases where the Van Velzen method is not applicable. The accuracy of estimation is of the same order as Van Velzen. The limitations of the method are,

- The method is less detailed than the Van Velzen method
- Applicable only in the range T_f to T_b
- No explicit treatment for heterocyclics or esters (apart from acetates).

The Morris method takes the following form

$$\ln(v/v^*) = 2.3026 J(1/Tr - 1)$$

$$J = (0.577 + \text{Sum}(D_i))0.5$$

The values of v^* are given for various categories of compounds. The constants v^* and the group contributions D are given in Morris.

8.5.3 Method of Letsou and Stiel

This is a corresponding states method with applicability over $0.76 < Tr < 1$. The method also predicts the viscosity at the critical point ($Tr = 1$). The accuracy is normally better than 5% up to $Tr = 0.92$ with higher errors encountered as the critical point is approached. Overall this is an excellent estimation method for high-temperature liquid viscosity. The only serious limitation is the restricted range of applicability.

The form of the relation is

$$v = (f_0 + w.f_1)/A$$

with

$$w = \text{acentric factor}$$

$$f_0 = a_0 + b_0Tr + c_0Tr^2$$

$$f_1 = a_1 + b_1Tr + c_1Tr^2$$

$$A = 0.176 \times 10^6 T_c^{0.1667} / M^{0.5} P_c^{0.667}$$

$$a_0 = 2.648 \quad a_1 = 7.425$$

$$b_0 = -3.725 \quad b_1 = -13.39$$

$$c_0 = 1.309 \quad c_1 = 5.933$$

In the above relations P_c is in bar and the viscosity is in units of Pa-sec.

8.5.4 Method of Przedziecki & Sridhar

In this method, the viscosity is related to changes in the specific volume.

$$v = V_0/E(V - V_0) \text{ centipoise}$$

$$V = \text{liquid molar volume in cc/mol}$$

$$E = -1.12 + V_c/D$$

$$D = 12.94 + 0.1 M - 0.23 P_c + 0.0424 T_f - 11.58 T_f/T_c$$

$$V_0 = 0.0085 w T_c - 2.02 + V_f / \{0.342(T_f / T_c) + 0.894\}$$

with

T_c = critical temperature, K

P_c = critical pressure, bar

V_c = critical volume, cc/mol

M = molecular weight

T_f = freezing point, K

w = acentric factor

V_f = specific volume at T_f

The authors recommend that the volumes be estimated from the Gunn and Yamada equation. The reader is referred to Reid for a discussion on this method. The method is less accurate below reduced temperatures of about 0.55. Errors vary widely but will normally be less than 20% for Tr greater than 0.55.

This method is used in CHEMPAK only where necessary. An error analysis by Reid et al indicates a higher level of error associated with this method than with the Van Velzen method for instance.

8.5.5 Interpolation and Extrapolation

Two regions are typically covered well by available experimental data, experimental correlations and by the above relations:

273 < T < 0.6 T_c: this region is normally covered by published data or by one of the methods 8.5.1, 8.5.2, 8.5.4

0.76 T_c < T < T_c: this region is well covered by the method of Letsou and Stiel (section 8.5.3)

This leaves two regions which are often not covered by the above methods

T_f < T < 273: this region may be covered by extrapolation using ln(v) versus 1/T extrapolation. The error due to the extrapolation in practise will not normally exceed 10% with a possible 20% error in the immediate vicinity of the freezing point.

0.6 T_c < T < 0.76 T_c: this region may be covered by interpolation between the 273 < T < 0.6 T_c region and the 0.76 T_c < T < T_c region using ln(v) versus 1/T interpolation. The errors due to interpolation in this case rarely exceed 5%.

8.5.6 The Effect of Pressure on Liquid Viscosity

The method of Lucas is applied:

$$\nu_L/\nu_{sL} = (1 + B.FA)/(1 + w.C.F)$$

ν_L = viscosity at pressure P

ν_{sL} = viscosity at saturation pressure P_s

$$F = (P - P_s)/P_c$$

w = acentric factor

$$A = 0.9991 - 0.0004674/(1.0523/Tr - 0.03877 - 1.0513)$$

$$B = 0.3257/(1.0039 - Tr) - 0.2906 - 0.2086$$

$$C = -0.07921 + 2.1616 Tr - 13.404 Tr^2 + 44.1706 Tr^3 - 84.8291 Tr^4 + 96.1209 Tr^5 - 59.8127 Tr^6 + 15.6719 Tr^7$$

8.6 Liquid Thermal Conductivity

8.6.1 Method of Latini et al

For specified categories of compounds, the method of Latini et al gives correlations for liquid conductivity for the range $Tr = 0.3$ to 0.8

The correlations are in the form

$$k = A(1 - Tr)^{0.38}/Tr^{0.167}$$

$$A = A_0 T_b^n M^m T_c^p$$

Category	A0	n	m	p
Alkanes	0.0035	1.2	-0.5	-0.167
Alkenes	0.0361	1.2	-1.0	-0.167
Cycloalkanes	0.0310	1.2	-1.0	-0.167
Aromatics	0.0346	1.2	-1.0	-0.167
Alcohols/Phenols	0.00339	1.2	-0.5	-0.167
Acids	0.00319	1.2	-0.5	-0.167
Ketones	0.00383	1.2	-1.0	-0.167
Esters	0.0415	1.2	-1.0	-0.167
Ethers	0.0385	1.2	-1.0	-0.167
Halides	0.494	0.0	-0.5	0.167
R20,R21,R22,R23	0.562	0.0	-0.5	0.167

Errors may be large for Diols and Glycols. The Acids equation is not applicable to Formic acid. The reader is referred to Reid for a discussion of the method.

8.6.3 Method of Sato-Riedel

This method gives the following relation:

$$k = (1.11/M^{0.5})f(Tr)/f(Tbr)$$

with

$$f(X) = 3 + 20(1 - X)^{0.667}$$

This method gives poor results for low molecular weight or branched hydrocarbons. Errors otherwise are likely to be less than 15%. The method should not be applied for $Tr > 0.8$

8.6.4 Method of Ely and Hanley

The method of Ely and Hanley has application to the high-temperature liquid region ($Tr > 0.8$). There are few data available for high temperature liquid conductivities. The method of Ely and Hanley is probably the best method available. Error estimates are unknown.

This method is used in CHEMPAK for $Tr > 0.8$ with caution. It appears to give reasonable results for non-polar compounds. Errors with polar compounds can be large.

8.6.5 The Effect of Pressure on Liquid Conductivity

The procedure derived from Missenard as presented in Danner and Daubert is employed:

$$k/k_s = 0.98 + 0.0079 Pr Tr^{1.4} + 0.63 Tr^{1.2} Pr / (30 + Pr)$$

k = conductivity at P

k_s = conductivity at P_s

8.8 Vapor Pressure

The vapor pressure is expressed in its reduced form

$$P_{vr} = P_v/P_c$$

Reduced vapor pressure varies from very low values at freezing point to unity at the critical point.

8.8.1 Published Correlations

The experimental correlations are commonly given in the following formats:

Wagner Equation

$$\ln(P_{vr}) = (aX + bX^{1.5} + cX^3 + dX^6)/T_r$$

with

$$X = 1 - T_r$$

FKT Equation

$$\ln(P_v) = a + b/T + c \ln(T) + dP_v/T^2$$

Antoine Equation

$$\ln(P_v) = a + b/(T + c)$$

8.8.2 Gomez-Thodos Vapor Pressure Equation

Gomez-Nieto and Thodos give the following equation:

$$\ln(P_{vr}) = B(1/T_{rm} - 1) + G(T_r^7 - 1)$$

$$G = aH + bB$$

$$a = (1 - 1/T_{br})/(T_{br}^7 - 1)$$

$$b = (1 - 1/T_{brm})/(T_{br}^7 - 1)$$

$$H = T_{br} \ln(P_c/P_b)/(1 - T_{br})$$

For non-polar compounds,

$$B = -4.267 - 221.79/(H_2.5 \exp(0.038 H_2.5)) + 3.8126/\exp(2272.33/H_3) + D$$

$$m = 0.78425 \exp(0.089315 H) - 8.5217/\exp(0.74826 H)$$

$$D = 0$$

except $D = 0.41815$ for He, 0.19904 for H₂, 0.02319 for Ne

For polar non-hydrogen-bonding compounds (e.g. ammonia and acetic acid),

$$m = 0.466 T_c^{0.1667}$$

$$G = 0.08594 \exp(0.0007462 T_c)$$

$$B = (G - aH)/b$$

8.8.2 Gomez-Thodos Vapor Pressure Equations

For polar hydrogen-bonding compounds (water, alcohols),

$$m = 0.0052 M_0.29 T_c^{0.72}$$

$$G = (2.464.M) \exp(0.0000098 MT_c)$$

$$B = (G - aH)/b$$

The advantages of this method are,

- fit guaranteed at $T = T_b$ and $T = T_c$
- good performance with polar compounds
- good performance over $T_r = 0.5$ to 1

In addition, tests carried out by Madison Technical Software show the clear superiority of this method especially at low temperatures over the Lee-Kesler method.

8.8.3 Lee-Kesler Vapor Pressure Equation

Lee and Kesler give the following vapor pressure equation:

$$\ln(P_{vr}) = f(0) + wf(1)$$

w = acentric factor

$$f(0) = 5.92714 - 6.09648/T_r - 1.28862 \ln(T_r) + 0.169347 T_r^6$$

$$f(1) = 15.2518 - 15.6875/T_r - 13.4721 \ln(T_r) + 0.43577 T_r^6$$

The characteristics of this equation are,

- guaranteed fit at $T_r = 1$ and 0.7
- accurate for non-polar compounds

This equation is used in the Lee-Kesler and Wu & Stiel equations of state.

8.8.4 Interpolation and Extrapolation

In many cases an accurate empirical equation is known which does not extend to the critical point or to the freezing point. The approach taken here is to fit the Wagner equation by least squares to the empirical equation and use the Wagner equation to extrapolate to the freezing point and to the critical point.

Extrapolation by this method to the critical method is a very accurate procedure. Extrapolation to the freezing point is less accurate but it does provide reasonable values.

In CHEMPAK, the vapor pressure correlations set out in this section are used to provide the basic data. Empirical relations are used wherever possible.

8.14 Notation

C Specific Heat

e Expansion Coefficient

h Enthalpy

\log Logarithm to base 10

\ln Natural Logarithm

m Dipole Moment

M Molecular Weight

P Pressure

R Gas Constant
r Riedel Parameter
s Entropy
T Temperature
v Viscosity
V Specific Volume
w Acentric Factor
x Mole Fraction
Y Wu & Stiel Polarity Factor
Z Compressibility

Subscripts

b Boiling
c Critical
es Equation of State
f Freezing
ig Ideal Gas
L Liquid
m Mixture
0 Low Pressure
p Constant Pressure
ra Rackett
ref Reference
r Reduced
s Saturated
v Vapor
v Constant Volume

Superscripts

(o) Simple Fluid
(r) Reference Fluid
(p) Polar Fluid

8.15 References

API Technical Data Book - Petroleum Refining, 4 Vols, API, Washington DC, 1988
CRC Handbook of Chemistry and Physics, CRC Press Boca Raton 1991

- Danner R P and Daubert T E , Data Prediction Manual, Design Institute for Physical Property Data , AIChE, NY 1983
- Daubert T E and Danner R P, Physical and Thermodynamic Properties of Pure Chemicals, Data Compilation, AIChE, Hemisphere NY 1989
- Engineering Sciences Data Units (ESDU), (9 Vols Data Compilation), London, England
- Gomez-Nieto M and Thodos G, Ind Eng Chem Fundam, Vol 17, 45, 1978, Can J Chem Eng, Vol 55, 445, 1977, Ind Eng Chem Fundam, Vol 16, 254, 1977
- Hankinson and Thomson, AIChEJ, Vol 25, 653, 1979
- International Critical Tables, National Research Council, 7 Vols, McGraw-Hill, NY 1926
- Joback K G, SM Thesis, MIT, June 1984
- Keenan et al, Steam Tables, John Wiley, NY 1978
- Knapp et al, Chem Data Ser, Vol VI, DECHEMA 1982
- Kreglewski and Kay, J Phys Chem, Vol 73, 3359, 1969
- Letsou A and Stiel L I, AIChEJ, Vol 19, 409, 1973
- Le Bas G, Molecular Volumes of Liquid Chemical Compounds, Longmans Green, NY 1915
- Lee B I and Kesler M G, AIChEJ, Vol 21, 510, 1975
- Li C C, AIChEJ, Vol 22, 927, 1976
- Lucas K, Chem Ing Tech, Vol 53, 959, 1981
- Missenard F A, Rev Gen Thermodyn, Vol 101, 649, 1970
- Morris P S, MS Thesis, Polytech Inst Brooklyn, NY 1964
- Perry et al , Chemical Engineer's Handbook (various editions), McGraw Hill, NY
- Plocker U J et al, Ind Eng Chem Proc Des Dev, Vol 17, 324, 1978
- Prausnitz and Gunn, AIChEJ, Vol 4, 430 and 494, 1958
- Reid R C et al, Properties of Liquids and Gases, 3rd Ed, McGraw Hill, NY 1977, 4th Ed, McGraw Hill, NY 1987
- Schick and Prausnitz, AIChEJ, Vol 14, 673, 1968
- Spencer and Danner, J Chem Eng Data, Vol 17, 236, 1972
- Spencer, Daubert and Danner, AIChEJ, Vol 19, 522, 1973
- Stiel and Thodos, AIChEJ, Vol 8, 229, 1962
- Teja A S et al, Chem Eng Sci, Vol 33, 609, 1978, AIChEJ, Vol 26, 337 & 341, 1980, Chem Eng Sci, Vol 36, 7, 1981, Ind Eng Chem Fundam, Vol 20, 77, 1981, Chem Eng Sci, Vol 37, 790, 1982, J Chem Eng Data, Vol 28, 83, 1983, Ind Eng Chem Proc Des Dev, Vol 22, 672, 1983
- Thomson, Brobst, Hankinson, AIChEJ, Vol 28, 671, 1982
- Tyn M T and Calus W F, Processing, Vol 21(4), 16, 1975
- Van Velzen D et al, Ind Eng Chem Fundam, Vol 11, 20, 1972
- Van Velzen et al, Liquid Viscosity etc, Euratom 4735e, Joint Nuclear Research Centre, ISPRA Establishment, Italy 1972
- Vargaftik N B, Tables on Therm Props Liq & Gases, 2nd Ed, Hemisphere, Washington DC, 1975

Wu G Z A and Stiel L I, AIChEJ, Vol 31, 1632, 1985

Yorizane et al, Ind Eng Chem Fundam, Vol 22, 454, 1983

Role of Pressure Junctions - Detailed Discussion

In modeling a generalized pipe network, it is possible to construct models that do not have a unique solution. A common occurrence of this is when a model contains one or more sections which are completely bounded by known flow rates. A series of examples with accompanying explanation will clarify situations where this occurs.

Before beginning, it will be helpful to consider an aspect of the philosophy of computer modeling. At the risk of stating the obvious, it must be recognized by the user that a computer model cannot calculate anything that cannot in principle be calculated by hand. All the computer does is accelerate the calculation process. Some users expect the model to generate independent information and become frustrated when the software requests additional information from them. But if the user was doing the calculation by hand that same information would still be required. The difference is that in the case of the hand calculation, the engineer would be forced to think through why the information is needed.

However, when the engineer interacts with AFT Fathom, they do not need to think at the same depth as in the hand calculation. So when AFT Fathom asks for the additional information, it is not as apparent why it is required. This disconnect in thinking can cause frustration for the user. Fully understanding the concepts in this topic will greatly reduce the frustration for some users, and will improve the quality of models for all users.

Examples

The simplest example is the system shown in Figure 1. In AFT Fathom terms, the system has two assigned flow junctions. Heat transfer has no affect on the point being made here and is therefore ignored in all examples.



Figure 1: Model with two assigned flows. This model does not have a unique solution.

Obviously, the flow in the pipe is known. But what is the pressure at the inlet? At the outlet? It cannot be determined because there is no *reference pressure*. The reference pressure is that pressure from which other pressures in the system are derived. There can be one or more reference pressures, but there always has to be at least one.

The model in Figure 1 can be built with AFT Fathom and if you try and run this model, AFT Fathom will inform you that it cannot run it because of the lack of a reference pressure. In AFT Fathom there are several junctions that can act as a reference pressure: the reservoir, assigned pressure, spray discharge, valve (with the exit valve option), orifice (with the exit orifice option), and exit type relief valves when open.

There are a couple other things worth noting about Figure 1. First is that the model in Figure 1 has redundant boundary conditions. When one flow is specified in a single pipe, the other end must have the

same flow. Thus the second known flow does not offer any new information. Because the conditions are redundant, there is no unique solution to the model in Figure 1. In any AFT Fathom model, there is always at least one unknown for each junction in the model – either pressure or flow rate. Sometimes there are two unknowns, such as a Pump. It is possible the user may not know either the flow or the pressure at the pump. In such cases the user is required to specify the relationship between flow and pressure. This relationship is called a pump curve, and by specifying the relationship between the two, effectively one of the two unknowns can be eliminated. Thus we always end up with one unknown for each junction.

Second, if the user was allowed to specify two flows as in Figure 1, they could specify them with different flow rates. Both ends of the pipe must have the same flow, so an inconsistency occurs. The basic reason the inconsistency is possible is because Figure 1 does not have a unique solution.

Figure 2 a-c shows three alternate configurations for the model in Figure 1. It is not possible to specify inconsistent conditions for any of the Figure 2 models, and they always have a unique solution no matter what input is specified by the user.

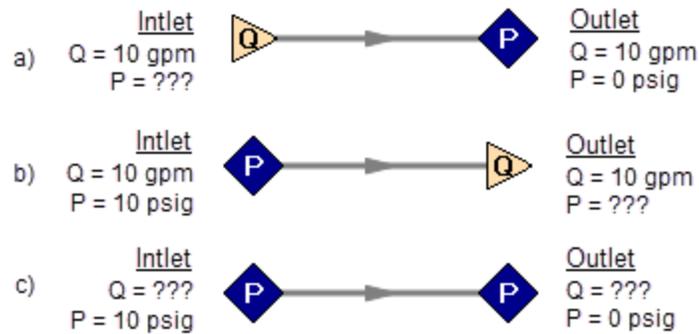


Figure 2a-c: Top two models with one pressure and one assigned flow, bottom model with two pressures. All of these models have a unique solution.

The four models in Figures 1 and 2 a-c contain all logical possibilities. In all four cases there are four things we want to know. The pressure and flow at the inlet, and the pressure and flow in the outlet. In each case we know two of these four. The problem is that in the Figure 1 model the lack of a reference pressure makes it impossible to determine the inlet and outlet pressures, even though we know the flow. The three models in Figure 2 a-c all have at least one pressure and thus all four of the desired parameters can be determined. Table 1 summarizes this. Also note in Table 1 that in the Figure 2c case the flow can be determined, but it is by iteration.

Table 1: Four logical possibilities of boundary conditions for single pipe systems

Case	Inlet Pressure	Inlet Flow	Outlet Pressure	Outlet Flow	OK?	Comments
Figure 1		X		X	NO	No reference pressure, cannot calculate pressures
Figure 2a		X	X		YES	1 known flow, can calculate everything directly
Figure 2b	X			X	YES	1 known flow, can calculate everything directly
Figure 2c	X		X		YES	2 pressures, can iterate for flow

As we consider multi-pipe systems, there are a host of other possibilities that present themselves. All other configuration possibilities which lack a reference pressure ultimately boil down to the same problem that exists in Figure 1.

The pipe lengths, diameters and friction factors are not included in the remaining models because they do not influence the main point of this topic. It is assumed that these parameters can be obtained for each pipe and the resulting pressure drop calculated with standard relationships.

Consider the model in Figure 3. With three boundaries having a known flow, this clearly has the same problem as the model in Figure 1. No unique pressures at any location can be calculated.

A pressure at any one of the three boundaries in Figure 3 would be sufficient to allow a unique solution of the system. It is also possible that there could be two pressures and one flow, or three pressures. As long as there is at least one pressure a unique solution exists.

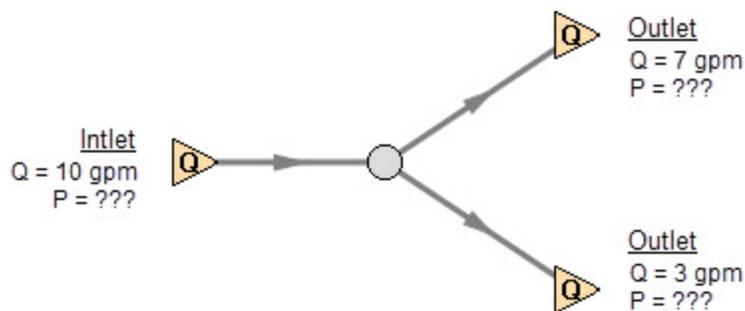


Figure 3: Three known flows at boundaries lack a reference pressure, similar to the model in Figure 1. This model does not have a unique solution.

Now consider the models in Figure 4. The Figure 4a model does have one reference pressure, but in this case there is still a problem. The Flow Control Valve (FCV) is controlling the flow to 10 gpm, and the downstream flow is demanding 10 gpm. The section of the system before the FCV can be solved (because there is a pressure upstream), but the section after the FCV does not have a unique solution. Why? In order to obtain a unique solution, the pressure drop across the FCV must be known. But any pressure drop could exist and satisfy the conditions of this model. It is thus not possible to determine the pressure at the outlet flow demand because it depends on the FCV pressure drop which is not known.

The model in 4b does have a pressure downstream of the FCV, and thus there is a unique pressure drop across the FCV and a unique solution exists for the model in Figure 4b.

If you input the remaining data for the Figure 4a model and run it in AFT Fathom, you will get the message shown in Figure 5. AFT Fathom identifies the part(s) of the model where a known pressure is needed.

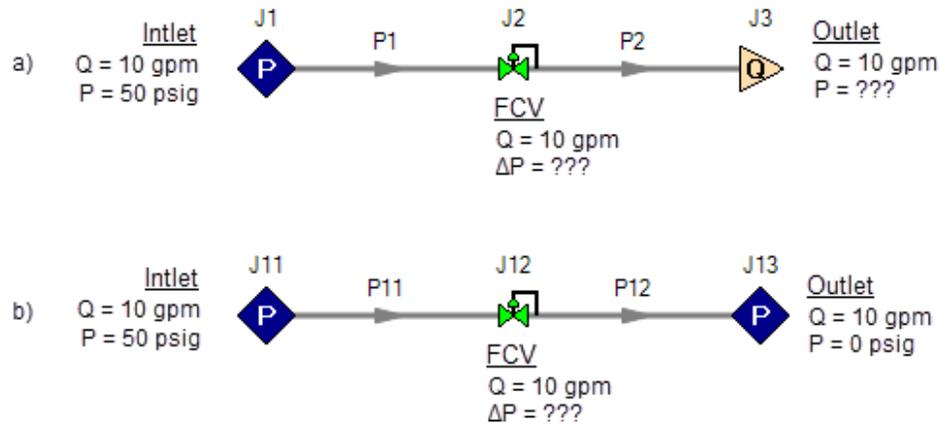


Figure 4a-b: The model at the top does not have a reference pressure after the Flow Control Valve so the pressure drop across the FCV cannot be determined. The top model does not have a unique solution. The bottom model has a pressure upstream and downstream, and a unique solution exists

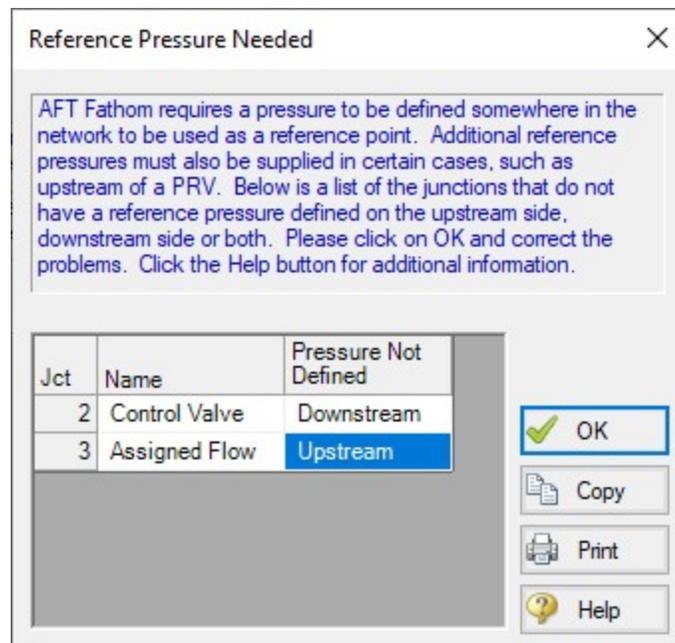


Figure 5: AFT Fathom message when you try to run the model shown in Figure 4a

Sizing Pumps with Flow Control Valves

An analogous situation to Figure 4a is when the user is trying to size a pump using the Pump with a fixed flow. The Pump with a fixed flow is really just a reverse FCV. It controls the flow by adding pressure, rather than reducing it. Figure 6 shows this case and is another example of a model without a unique solution, requiring more than one pressure junction. If the user instead entered a pump curve for this model, it could be solved.

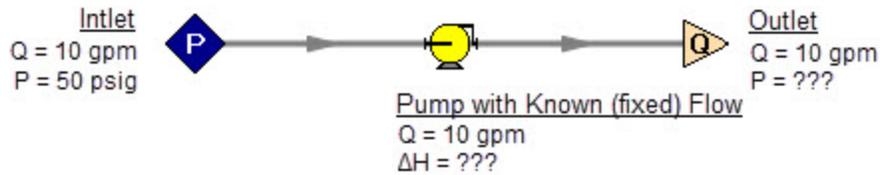


Figure 6: A pump modeled as a fixed flow behaves identically to the FCV model in Figure 4a, for which no unique solution exists

Increasing in model complexity, Figure 7 shows some additional examples of models without a unique solution. In particular, Figure 7b has a pump with a fixed flow that can add as much pressure as it wants, and the downstream FCV can then take out as much as it wants. There are an infinite number of possible solutions.

While it would be highly unusual for a system to have two FCV's in series, it is worth considering the model in Figure 7b further. It is common to have an FCV in series with a pump. How does one size the pump in such a case? There are several ways to do this, one of which is to change the pump from a fixed flow to an actual pump curve. Trying multiple pump curves will guide you to the best pump.

But there is a better way. Consider for a moment the pressure drop across the FCV in Figure 7b. In the installed system, is any pressure drop acceptable? Usually not. It is typical to have pressure drop limits based on the system design and the FCV itself. A common requirement is a minimum pressure drop across the FCV. If no minimum pressure drop exists, let's say we choose a pump that results in the pressure drop across the FCV being 0.01 psid. If the system is built and the pump slightly underperforms its published curve, the FCV will not be able to control to its flow control point. Even if the pump does operate exactly on the published curve, eventually fouling in the pipes will result in increased pipe resistance (and pressure drop) and again the FCV will not be able to control to its flow control point.

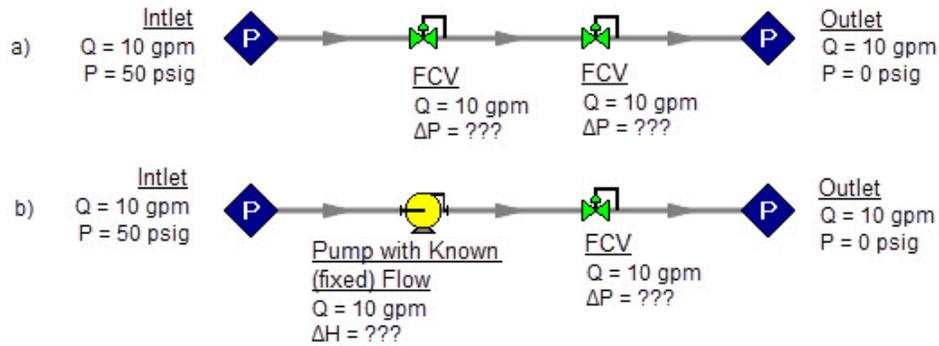


Figure 7: Neither model above has a unique solution. The top model has two flow control valves in series. The bottom model has a pump modeled as a fixed flow in series with an FCV. In both cases either a third pressure junction is needed between the two middle junctions, or one of the junctions must be changed from a flow controlling device.

To avoid these kinds of problems in installed systems, a minimum Cv or pressure drop across the FCV is typically required. This minimum pressure drop provides the key to sizing the pump in Figure 7b. Rather than model the FCV as a flow control valve, instead model it as a pressure drop control valve (PDCV) set to the minimum pressure drop requirement. Then a unique solution exists, the model will run and the pump can be sized. Once the pump is sized and an actual pump curve exists, the pump curve can be entered for the pump and the valve can be switched back to an FCV. Remember that when we use the PDCV, we are still getting the flow we want for the true FCV, because the pump is now providing the control.

Think for a moment what we just did. We went through a thought process that many engineers have gone through during their hand calculations. Specifically, we size the pump such that the FCV pressure drop is minimized. If the FCV has a larger pressure drop than the minimum, the pump must pump harder to overcome this excess pressure drop. In short, it must use more energy and is thus less efficient. Figure 8 depicts this process.

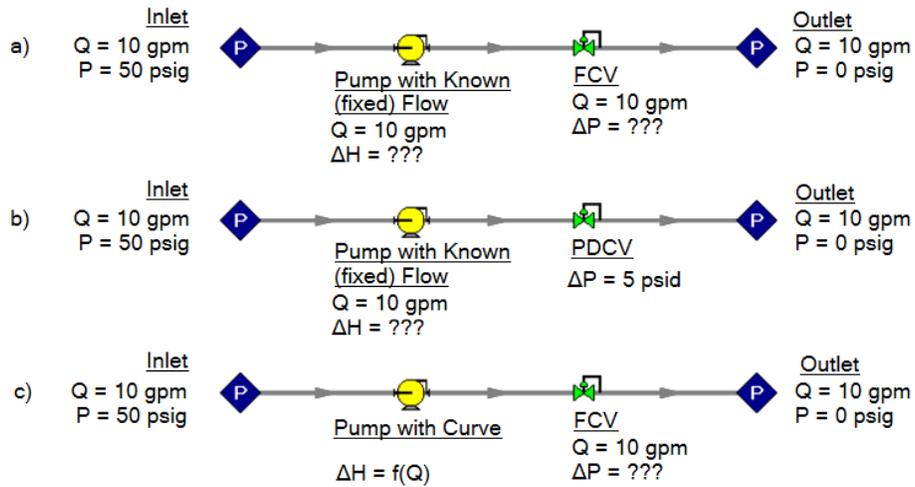


Figure 8a-c: The model at the top is the same as Figure 7b and does not have a unique solution. To size the pump, change the model to the second one shown above, which uses a pressure drop control valve (PDCV) rather than an FCV. Once the second model is run, the pump is sized, a pump curve exists, and the model at the bottom can be run using an FCV.

Let's make a slight addition in complexity to the model in Figure 8. In this case we have two flow control valves in parallel (Figure 9). To size this pump, we apply the process described in Figure 8. We choose one of the FCV's, make it a PDCV, size the pump, choose a pump with an actual pump curve and enter it into the model, then return the PDCV to an FCV.

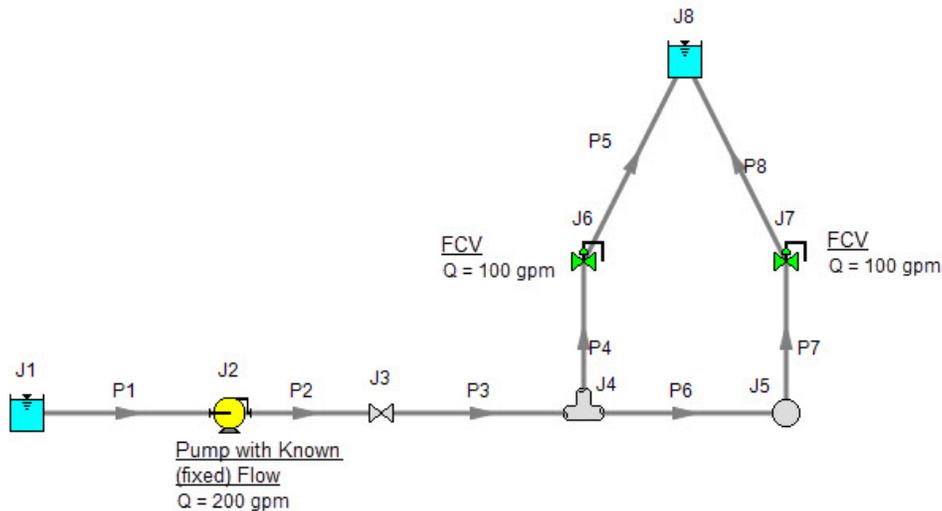


Figure 9: A pump with a fixed flow in series with two parallel FCV's. The model in its current form does not have a unique solution. To size the pump, first change the FCV at J7 to a PDCV.

This brings up the question, which FCV should one choose to turn into a PDCV? When FCV's are put in parallel, frequently the pipe design has one of the FCV's further away from the pump than the others.

Because of the additional piping leading to this FCV, it will be the weakest link in the chain of parallel FCV's by virtue of having the lowest pressure drop across it. The most remote FCV should be chosen as the PDCV. If the most remote FCV is chosen, when the minimum pressure drop required is applied to the PDCV all other FCV's in the parallel system will have greater than the minimum pressure drop — thus satisfying the pressure drop requirement for those FCV's as well.

What if you do not know which FCV is the most remote? In this case make your best guess, change it to a PDCV at the minimum required pressure drop, run the model, then verify whether all other FCV's have a pressure drop that meets or exceeds the requirement. If not, then the FCV with the smallest pressure drop as determined by the first run is in reality the weakest FCV. Choose this FCV, change it to a PDCV, and then change the original PDCV back to an FCV since it is not the weak link. If the pipe system leading to each FCV is truly identical and all FCV's have identical pressure drops, any of the FCV's will serve as the PDCV.

The preceding process can be extended to cases where there are three or more FCV's in parallel, and also cases with more than one pump in parallel supplying the FCV's.

In Figure 9 the most remote valve is J7, the one on the right. If the minimum pressure drop is 5 psid, change J7 to a PDCV with a 5 psi drop. This results in the output shown in Figure 10a. Notice how the pressure drop across the other FCV at J6 (6.837 psid) is greater than 5 psid because it is closer to the pump. Also notice how the flow through J7 is still 100 gpm even though it is not controlling flow. Once again, the reason for this is that the pump is controlling the flow to 200 gpm, and with the J6 FCV controlling to 100 gpm, the excess must go through J7. The pump is thus sized and its pump pressure rise is shown in Figure 10b (127 ft.).

General Warnings Design Alerts Pump Summary Valve Summary Reservoir Summary										
Jct	Name	Valve Type	Vol. Flow (gal/min)	Mass Flow (lbm/sec)	dP Stag (psid)	dH (feet)	P Static In (psia)	Cv	K	Valve State
10	Valve	REGULAR	200.0	27.76	1.229	2.841	66.33	180.28	0.5000	Open
4	Control Valve	FCV	100.0	13.88	6.489	14.997	55.34	39.23	10.5569	Open
7	Control Valve	PDCV	100.0	13.88	5.000	11.555	53.85	44.70	8.1340	Open

General Warnings Design Alerts Pump Summary Valve Summary Reservoir Summary													
Jct	Results Diagram	Name	Vol. Flow (gal/min)	Mass Flow (lbm/sec)	dP (psid)	dH (feet)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (hp)	BEP (gal/min)	% of BEP (Percent)	NPSHA (feet)	NPSHR (feet)
2	Show	Pump	200.0	27.76	54.78	126.6	100.0	N/A	6.390	N/A	N/A	48.04	N/A

Figure 10a-b: Results from changing J7 in Figure 9 to pressure drop control valve

Using Fixed Head Rise Pumps

An alternate method of sizing pumps with FCV's is to use the fixed head rise pump model. Here the pump head can be adjusted while the flow control valves can be modeled as FCV's. This approach is particularly powerful when used with the [GSC Module](#). This would entail specifying the pump head rise as the variable at the min Cv or pressure drop across the control valve as the goal. If there are multiple, parallel control valves, then a group max/min can be used.

Pressure Control Valves

The problem of non-unique solutions can also occur when a Pressure Reducing Valve (PRV) or Pressure Sustaining Valve (PSV) is used. A PRV is used to control pressure downstream of the valve, while a PSV controls pressure at the valve inlet. One thing that PRV's and PSV's have in common with FCV's is that the pressure drop cannot be known ahead of time. The pressure drop depends on the balance of the pipe system.

Consider the simple system shown in Figure 11. With the pressure controlled downstream of the PRV, the pressure upstream is not known. Without knowledge of the PRV's pressure drop, the pressure upstream of the PRV cannot be determined and thus no unique solution exists.

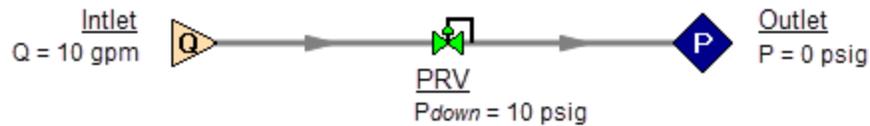


Figure 11: If a PRV is downstream of a known flow the PRV pressure drop cannot be calculated. This model does not have a unique solution.

Figure 12 shows the possible cases with PRVs and PSVs, while Table 2 comments on the six cases. In summary, at least one known pressure is always needed on the side of a pressure control valve opposite of the controlled side.

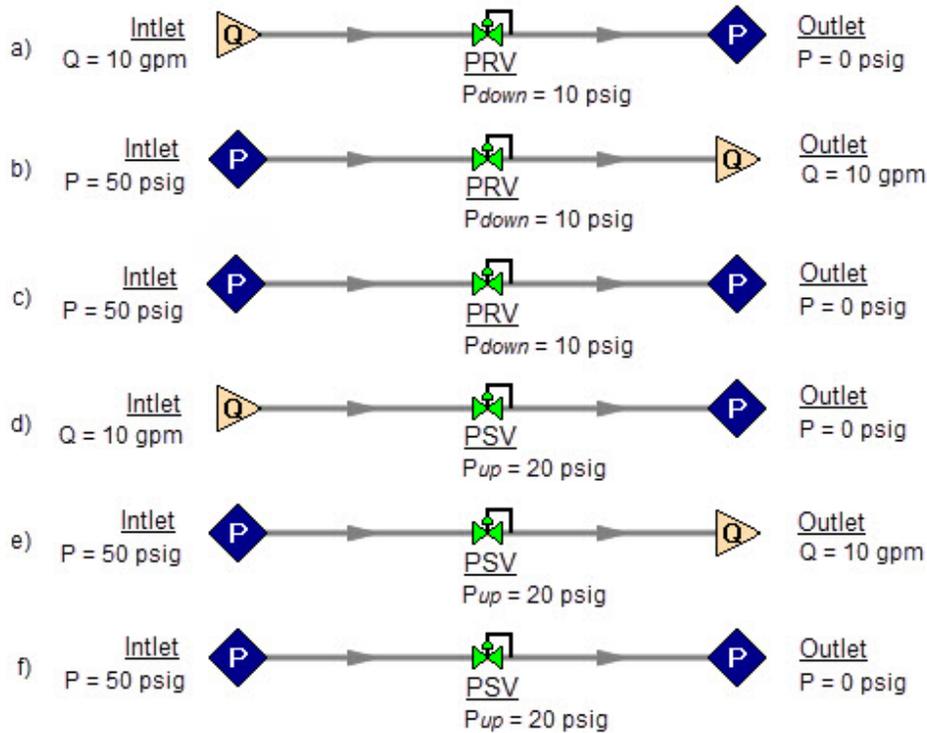


Figure 12a-f: Six possibilities for PRV and PSV configurations. Cases A and E do not have unique solutions. See Table 2 for comments.

Table 2: Summary of six possibilities for PRV and PSV configurations.

Case	Inlet Pressure	Inlet Flow	Outlet Pressure	Outlet Flow	OK?	Comments
Figure 12a - PRV		X	X		NO	No reference pressure upstream of PRV, cannot calculate
Figure 12b - PRV	X			X	YES	1 known flow, can calculate everything directly
Figure 12c - PRV	X		X		YES	2 pressures, can iterate for flow
Figure 12d - PSV		X	X		YES	1 known flow, can calculate everything directly
Figure 12e - PSV	X			X	NO	No reference pressure downstream of PSV, cannot calculate
Figure 12f - PSV	X		X		YES	2 pressures, can iterate for flow

If a pump is being sized in series with a PRV or PSV, modeling the pump as a fixed flow will not permit a unique solution if the pump is upstream of a PRV or downstream of a PSV. The reasons for this are the same as those in the previous section on FCV's.

In combination with the previous examples, it should be apparent how the non-unique solution problem can occur in more complicated systems. In all such cases, the reasoning will boil down to the same problems already discussed.

Closing Parts of a System

A model may be built which has sufficient information to obtain a unique solution. Then a user closes a pipe or junction and all of a sudden the problem of no unique solution occurs. This is shown in Figure 13. The top model has a unique solution and will run fine. In the bottom model the valve is turned off and the section of the system downstream of the valve has only known flows. No reference pressure exists in this section because it is isolated from the known pressures in the section on the left.

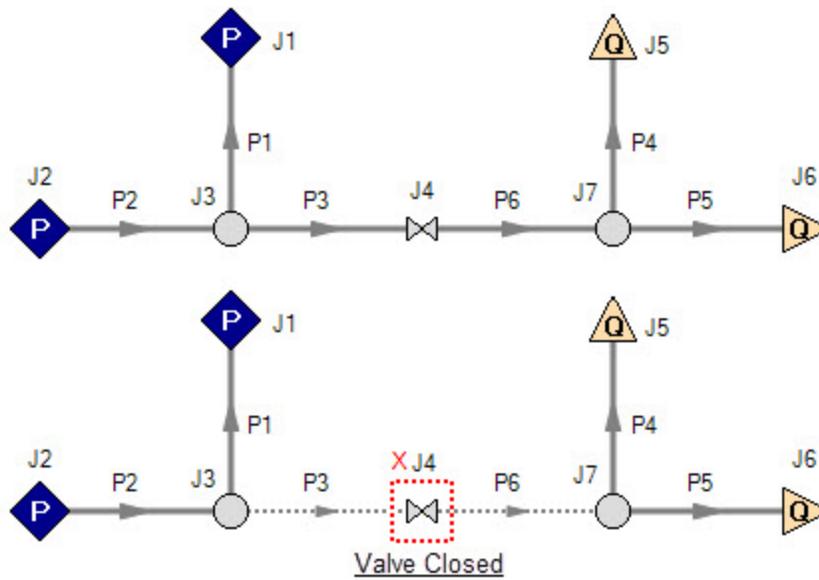


Figure 13: Top model has reference pressures at J1 and J2. Bottom model has closed valve at J4 which isolates the J5 and J6 assigned flows. There is no reference pressure for J5 and J6 and no unique solution exists for the bottom model.

Summary

AFT Fathom models always require one reference pressure, and may require more if the engineer uses control valves, pumps with fixed flows, or closed pipes or junctions. The actual number of reference pressures needed depends on the model, but AFT Fathom always checks to be sure sufficient reference pressures exist and stops to warn the user when they do not.

This capability is a powerful diagnostic feature that will help guide the user in building meaningful models.

Related Topics

Role of Pressure Junctions and How They Work	72
--	----

Related Examples

Pump Sizing and Selection with Flow Control Valves	882
--	-----

Pipe Material Library References

Prior to AFT Fathom 9, pipe material properties were compiled from a variety of unofficial sources. Many of the original sources are now obsolete. Beginning in AFT Fathom 9, pipe material properties are taken directly from national/international standards such as: ASTM, ANSI, AWWA, and ASME. Several new material standards are being added. For example: DIN and JIS. With the addition of a significant number of new pipe material standards, the increased volume and scope necessitate splitting the pipe material library into several smaller independent libraries. All of the pipe material found in AFT Fathom 7 is still available, but is now called "AFT Customary".

Maintaining several independent pipe libraries facilitates including or excluding specific standards. For example: a pipe engineer in Germany may prefer to use only ISO and DIN standards. The [Library Browser](#) can be used to "Connect" or "Disconnect" each of the pipe material libraries. A list of available pipe libraries can be found below.

Note that while AFT attempts to provide this standard information to make defining pipe information easier for the user, the prudent engineer will always verify this information against manufacturer specifications. ASTM and ANSI standards exist for the thickness and diameter specifications for some of these materials. Furthermore, AFT attempts to source reference information from reputable and authoritative sources. However, data for pipe roughness and thermophysical properties including density, thermal conductivity, modulus of elasticity, and Poisson's ratio have no single standard to follow. Thus, the user should always check that the pipe material inputs match the actual pipe material data for their system.

- **Copper Pipe - ASTM -**
 - Source = ASTM B42-10
 - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2; Perry's chem eng Handbook, 7th ed. Table 10-33
- **Copper Tubing - ASTM -**
 - Source = ASTM B88-09
 - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2; Perry's chem eng Handbook, 7th ed. Table 10-31; www.cranecopper.com.au/sizes/astmb88.asp
- **Ductile Iron - ANSI -**
 - Source = ANSI-AWWA C150-A21.50-08
 - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2
- **Fiberglass - Green Thread -**
 - Source = Fiber Glass Systems product data, http://www.nov.com/Segments/Completion_and_Production_Solutions/Fiber_Glass_Systems/Chemical_Industrial/Green_

- Thread.aspx
- Roughness References = Fiber Glass Systems product data
- Density References = Average value calculated based on ID, OD and Weight in Fiber Glass Systems product data
- Thermal Conductivity References = Fiber Glass Systems product data
- Modulus of Elasticity References = Fiber Glass Systems product data
- Poisson's Ratio References = Fiber Glass Systems product data
- Additional References = ASTM D2996 - 17
- **HDPE - AFT Customary -**
 - Source = Applied Flow Technology
 - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2
- **PVC - ASTM -**
 - Source = ASTM D1785-06
 - Additional References = Thermal data from: Materials Science and Engineering, Callister, Table C.4
- **PVDF - AFT Customary -**
 - Source = George Fischer Engineering handbook page 425
 - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2
- **Rectangular Duct - AFT Customary -**
 - Source = Applied Flow Technology
- **Stainless Steel - ANSI -**
 - Source = ASME-B36_19M_2004
 - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
 - Thermal Conduct References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Type 304
 - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
 - Poisson's Ration Referenecs = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for Type 304 @ 20oC
- **Stainless Tubing - AFT Customary -**
 - Source = Applied Flow Technology
 - Additional References = Thermal conductivity data from: Piping Handbook, Nayyar, 7th ed, Table A3.2
- **Steel - AFT Customary -**
 - Source = Applied Flow Technology
 - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
 - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3

- Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
- Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
- **Steel - ANSI -**
 - Source = ASME B36.10-2004
 - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Commercial Steel
 - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
 - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
 - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
 - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
 - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - DIN 2391-1 -**
 - Source = DIN 2391-1-1994
 - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Commercial Steel
 - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
 - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
 - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
 - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
 - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - DIN 2458 -**
 - Source = DIN 2458-1981
 - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Commercial Steel
 - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
 - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
 - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
 - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
 - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10216-1 (s1) -**
 - Source = BS EN 10216-1-2002
 - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Commercial Steel

- Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
- Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
- Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
- Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
- Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10216-2 (s1) -**
 - Source = BS EN 10216-2-2002
 - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
 - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
 - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
 - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
 - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
 - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10217-1 (s1) -**
 - Source = BS EN 10217-1-2002
 - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
 - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
 - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
 - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
 - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
 - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10217-2 (s1) -**
 - Source = BS EN 10217-2-2002
 - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
 - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
 - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
 - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
 - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
 - Additional References = ANSI 36.10, CRANE Technical Paper 410

- **Steel - EN 10217-4 (s1) -**
 - Source = BS EN 10217-4-2002
 - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
 - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
 - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
 - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
 - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
 - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10217-5 (s1) -**
 - Source = BS EN 10217-5-2002
 - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
 - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
 - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
 - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
 - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
 - Additional References = ANSI 36.10, CRANE Technical Paper 410
- **Steel - EN 10217-6 (s1) -**
 - Source = BS EN 10217-6-2002
 - Roughness References = CRANE Technical Paper 410, 1988 edition, A-23 for Comerical Steel
 - Density References = Piping Handbook, Nayyar, 7th ed, Mohinder L Nayyar, Table A3.3
 - Thermal Conductivity References = Piping Handbook, Nayyar, 7th ed, Table A3.2 for Wrought Carbon Steel, 0.23 C, 0.635 Mn
 - Modulus of Elasticity References = Piping Handbook, Nayyar, 7th ed, Table A3.1
 - Poisson's Ratio References = Perry's Chemical Engineer's Handbook, 7th ed, 1997 p.28-45 Table 28-18 for SA-35P12 @ 20oC
 - Additional References = ANSI 36.10, CRANE Technical Paper 410

Related Topics

Libraries.....	707
Library Manager.....	708
Library Browser.....	708
Network Library Overview.....	723
Library Verification.....	727

Library Locations in General Preferences	729
Fluid Library.....	713
Pipe Material Library.....	714
Junction Library.....	712
Insulation Library.....	717
Fittings & Losses Library.....	717

Output Messages & Warnings

Various types of messages may be given in the [output](#) when you run a model. These messages are intended to alert you about problems with the model or solution. There are different message classifications depending on the severity, but all message types should be reviewed when they are displayed in the output.

The links below will take you to a message list for the corresponding message type with a brief explanation given for each.

1. [CRITICAL WARNING](#) - **Very High Severity** - The model is not accurately predicting behavior, and the solution may violate the [engineering assumptions](#) of the software. These issues must be resolved for the model to be trustworthy, and the output should only be used for error resolution or troubleshooting purposes and never for engineering decisions.
2. [WARNING](#) - **High Severity** - A serious issue with the model or results exists. For example, the solution has exceeded a valid data range, or potentially unrealistic behavior is occurring. A real system may encounter operational problems under the conditions.
3. [CAUTION](#) - **Moderate/Low Severity** - Something may be wrong with the model or of interest for the user to double check. Depending on the application, the results may or may not be affected. If results are affected, it would be to a lower degree compared to a Warning.
4. [INFORMATION](#) - An Information message is intended to make the user aware that an advanced feature is being used in the model, or to provide comments or suggestions relating to an advanced feature. Following suggestions may achieve higher accuracy results if the right conditions apply, but can also cause incorrect results if not applied properly. If the user understands the feature then no action is required.

Any message that directly lists a pipe/junction can be double-clicked to take you directly to the object on the Workspace. Whenever a Warning or Critical Warning message type is displayed, the General Results section will display red text.

Related Topics

Troubleshooting	797
Critical Warnings	772
Warnings	776
Cautions	784
Information	793

Critical Warnings

The model is not accurately predicting behavior, and the solution may violate the [engineering assumptions](#) of the software. These issues must be resolved for the model to be trustworthy, and the output should only be used for error resolution or troubleshooting purposes and never for engineering decisions.

CRITICAL WARNINGS

FCV/PRV/PSV could not control pressure - Pressure was added to achieve control flow/pressure

A [Critical Warning message](#) in AFT Fathom

Full Message Text

Several variations are possible depending on which Valve Type is selected in the Control Valve Properties window:

- FCV could not control flow at Junction X - Pressure was added to achieve control flow. See Valve Summary for more information.
- PRV could not control pressure at Junction X - Reverse flow through valve, pressure was added to achieve control pressure. See Valve Summary for more information.
- PSV could not control pressure at Junction X - Pressure was added to achieve control pressure. See Valve Summary for more information.

Explanation

The Control Valve could not control to its set point and the [Always Control \(Never Fail\) setting](#) is enabled in the Control Valve Properties window. When the Always Control setting is enabled the program has the freedom to add pressure in order to make the solution mathematically possible. Then, the Valve Summary in the [Output window](#) gives more information about exactly how much pressure was added via the Pressure Drop Stagnation (dP) or Head Loss (dH) parameters. A negative dP or dH means it was a pressure rise. Normally you would see a positive dP or dH meaning it was a pressure loss. Furthermore, the Valve State output parameter in the Valve Summary will show "Failed - Had to add pressure".

Required Action

If you want the Control Valve to fail open or closed, then you can disable the Always Control setting in the [Control Valve Properties window](#). Whether the valve fails open or closed is dictated by the Use Default Actions setting and related options below it.

If you intended for the Control Valve to actively control, then this critical warning identifies that the pressure is too low at this location and the Control Valve cannot meet the set point. This could indicate a mistyped input or an operational problem. Double-check the inputs and corresponding units for the model including control valve settings, pumps, pipe sizes, and fluid properties. If there are no input typos, then there are many different conclusions that could be drawn, each one requiring a different solution approach that will depend on your unique system: undersized pump, higher than expected junction losses or K Factors, higher than expected pipe friction, pipe fouling changing the internal diameter/velocity, or [Laminar and ATKF corrections](#) causing higher losses.

Global energy balance disabled by user in Advanced Parameters Panel

A [Critical Warning message](#) in AFT Fathom

The displayed results do not represent a valid converged solution because the Energy Balance option was disabled by the user on the [Advanced Parameters panel](#) in the [Analysis Setup window](#).

Related Topics

Advanced Parameters Panel.....	179
Global mass balance disabled by user in Advanced Parameters Panel.....	774

Global mass balance disabled by user in Advanced Parameters Panel

A [Critical Warning message](#) in AFT Fathom

The displayed results do not represent a valid converged solution because the Mass Balance option was disabled by the user on the [Advanced Parameters panel](#) in the [Analysis Setup window](#).

Related Topics

Advanced Parameters Panel.....	179
Global energy balance disabled by user in Advanced Parameters Panel.....	774

Junction X had reverse flow - Pump head could not be predicted

A [Critical Warning message](#) in AFT Fathom

This message displays for centrifugal pumps when the flowrate goes backwards through the pump. The pump head rise under such conditions is unknown. The dead head value is used.

Pipe sequences of zero friction exist, flow rate cannot be calculated

A [Critical Warning message](#) in AFT Fathom

This message is displayed if all of the pipes in a flow path are defined as frictionless by using the zero-length connector option, or the "frictionless" friction model.

At least one pipe in every flow path needs to include frictional losses in order to perform hydraulic calculations.

Setting relaxation lower than relative pressure/flow tolerance gives unreliable results

A [Critical Warning message](#) in AFT Fathom

Because of the nature of relaxation and relative solution tolerance, setting the relaxation to a value close to or lower than relative tolerance can lead to false convergence and invalid results. AFT Fathom alerts the user to this when inputting data in the [Tolerance panel](#) or [Iterations panel](#) in the [Analysis Setup window](#), and also alerts the user once the solution has been obtained.

There were static pressures below absolute zero in the model

A [Critical Warning message](#) in AFT Fathom

Full Message Text

There were static pressures below absolute zero in the model. While this is mathematically correct, it is not physically realistic. Please check the model.

Explanation

The Solver is mathematically capable of calculating negative absolute pressures which satisfy the [fundamental equations](#), but this of course is not physically realistic. This message notifies the user of this unrealistic situation so that appropriate changes can be made to the model. If the model is based on a real functioning system, then all input data should be double checked. Furthermore, it may be possible that the real system is experiencing cavitation where the static pressure is dropping below the vapor pressure of the fluid at one or more locations. There could be other discrepancies such as worn components, pumps with lower efficiency, or fouling of components that affects the internal pipe areas, velocities, and pressure drops.

Required Action

The user must either correct the model such that this message is not displayed, or understand that all results are not physically possible and the current configuration of the model is not achievable in reality. Something in the model is causing the pressures to be too low which could be from any number of reasons:

- A pipe or junction is providing too much pressure drop.
- The flow demand from an Assigned Flow or Control Valve junction is too high causing too much friction loss.
- The [boundary conditions](#) do not provide enough pressure differential.
- [Laminar corrections](#) or Non-Newtonian viscosity models may be causing large pressure drops.

One of the best ways to start diagnosing the root cause is to review the [Model Data](#). If all inputs for Analysis Setup, pipes, and junctions has been verified, it is recommended to make various [Profile Graphs](#) to visualize how the pressure changes throughout the various flow paths of the model. You may find a specific junction or pipe that is causing the pressure to drop below absolute zero.

These results do not represent a converged solution

A [Critical Warning message](#) in AFT Fathom

This message alerts you to the fact that you chose to have results displayed for an unconverged solution. You should only use the displayed results for purposes of determining the reasons for nonconvergence. Under no circumstances should you actually use any of the results for engineering purposes.

Related Topics

Troubleshooting 797

Weir Liquid Level Convergence Problem

A [Critical Warning message](#) in AFT Fathom

This message indicates that the flow required at the weir to maintain flow continuity in the system is higher than the maximum flow the weir can deliver. It is recommended to review the system and weir inputs to resolve the issue.

Warnings

A Warning is of High Severity, and should always be reviewed before considering the results to be meaningful for the system.

A serious issue with the model or results exists. For example, the solution has exceeded a valid data range, or potentially unrealistic behavior is occurring. A real system may encounter operational problems under the conditions.

WARNINGS

Cavitation May Exist at Junction X

A [Warning message](#) in AFT Fathom

The local static pressure at this junction was obtained using the [Restricted Area](#), and it is below the vapor pressure of the fluid.

Chempak temperature and or pressure is out of range for the following

A [Warning message](#) in AFT Fathom

The final solution had temperature and/or pressure conditions in the specified pipes obtained with [Chempak physical property data](#) which were beyond Chempak's range of applicability. The range value is also displayed. This indicates that the results are probably not valid. It also may indicate an input error in the model because of the extreme temperature or pressure state.

Control valve open percentage outside valid range of 0-100% at Junction X

A [Warning message](#) in AFT Fathom

This is a cautionary notice that a [control valve](#) which includes open percentage data is operating outside the range of 0-100%. This could indicate an error in input, or an improperly sized valve.

Energy may not be balanced at Junction X (total= Y)

A [Warning message](#) in AFT Fathom

This warning alerts you to the fact that the converged energy balance results may be incorrect. Once a converged solution is obtained, AFT Fathom checks all junctions to ensure an energy balance exists. It compares the imbalance to the mass flow weighted enthalpy [tolerance](#) you specified, and when it appears to exceed the tolerance, it gives you this warning.

If the magnitude of the out of balance flow is close to the tolerance you specified, then it is generally safe to ignore this warning, although you may want to tighten your tolerances further and rerun the model.

If the magnitude of the energy out of balance is large, then you have a converged solution that is incorrect. This can occur in rare circumstances. When this happens, then it is usually an indicator that your model is not realistic.

You should always review each of these warnings in detail, and perform hand calculations on the energy balance at the affected junctions to verify whether an acceptable solution exists.

Note: This warning will usually occur when you assign isothermal boundary conditions to pipes in your model. If pipes are forced to flow at the isothermal temperature you specify, it is usually not possible to balance energy at the junctions. The warning can be ignored in such circumstances, because the energy imbalance is a natural result of the isothermal assumption.

Related Topics

Tolerance Panel.....	163
Energy Equation at a Branch.....	737
Verifying Network Solutions.....	740

FCV could not control flow at Junction X

A [Warning message](#) in AFT Fathom

This warning alerts you that the [Flow Control Valve](#) at junction X [was not able to control the flow](#), and backflow may have occurred.

Flow is into system at Reservoir Junction X

A [Warning message](#) in AFT Fathom

Full message is:

"Flow is into system at Reservoir Junction X. Pipe intake elevation above the surface will supply gas, not liquid."

When a pipe exits above a [reservoir](#) junction liquid level it is only meaningful if it is discharging into the reservoir. If the pipe becomes an intake, it will not supply liquid but whatever vapor is above the reservoir liquid. This warning alerts you to the fact that the reservoir had a pipe above liquid level that was taking in fluid. AFT Fathom assumes the intake fluid is the same as the reservoir fluid, but that is likely to not be the case. Results are likely invalid.

Flow may not be balanced at Junction X (total= Y)

A [Warning message](#) in AFT Fathom

This warning alerts you to the fact that the converged results may be incorrect. Once a converged solution is obtained, AFT Fathom checks all junctions to ensure a flow balance exists. It compares the imbalance to the flow [tolerance](#) you specified, and when it appears to exceed the tolerance, it gives you this warning.

If the magnitude of the out of balance flow is close to the tolerance you specified, then it is generally safe to ignore this warning, although you may want to tighten your tolerances further and rerun the model.

If the magnitude of the flow out of balance is large, then you have a converged solution that is incorrect. This can occur in rare circumstances. When this happens, it is usually an indicator that your model is not realistic.

You should always review each of these warnings in detail, and perform hand calculations on the flow balance at the affected junctions to verify whether an acceptable solution exists.

Related Topics

Tolerance Panel.....	163
Continuity Equation at a Branch.....	737
Verifying Network Solutions.....	740

Flow rate at drop off point at Junction X

A [Warning message](#) in AFT Fathom

This warning is for [jet pumps](#), and informs the user that the flow rate is at the edge of the curve. Results may not be valid.

GSC Goal may not be valid. User goal = X. Final goal result = Y.

A [Warning message](#) in AFT Fathom

([GSC Module Only](#))

This is a warning message that appears when the [GSC module](#) search was not able to meet a [goal](#). The solution may still be acceptable if the final goal result is reasonably close to the goal.

HGL, EGL and head loss results may not be meaningful for variable density systems

A [Warning message](#) in AFT Fathom

The concepts of HGL (Hydraulic Gradeline) and EGL (Energy Gradeline) are only valid for constant density systems. This warning is given when HGL, EGL or head loss results are displayed in the output and the system has [variable density](#). It alerts the user that HGL/EGL and head loss output may not be meaningful. Unless otherwise noted, all other output is valid.

This warning can be hidden by setting all head results to not be shown using [Output Control](#).

Junction X %BEP (Y%) is above/below the POR range

A [Warning message](#) in AFT Fathom

The calculated %BEP of Pump X is outside of the POR range as defined in [Pump Configuration](#).

Junction X exceeded runout flow rate

A [Warning message](#) in AFT Fathom

[Centrifugal pump curves](#) typically have a point at which they no longer operate on the curve. This may be called "end of curve" or "runout". Data defining runout for a particular pump is given by the manufacturer. Such data has been defined for this pump in the Pump Configuration window [Configuration Data tab](#), and the system is operating past the runout flow rate. Thus the results are invalid, because the pump likely cannot actually operate at that point.

Junction X flow (Y gal/min) is above/below the AOR range (Z gal/min)

A [Warning message](#) in AFT Fathom

The calculated flow of Pump X is outside of the AOR range as defined by the user in [Pump Configuration](#).

Junction X has insufficient pump NPSH/NPSP...

A [Warning message](#) in AFT Fathom

The available pump NPSH (or NPSP) is less than the required NPSH defined for the [pump](#). The pump will cavitate if changes are not made to increase the NPSH available at the pump.

Negative added pressure by pump at Junction X

A [Warning message](#) in AFT Fathom

Full Message text is:

"Negative added pressure by pump at Junction X."

A pump junction with a fixed flow resulted in a pressure loss at the [pump junction](#). This warning is meant to alert the user that the pump is not adding pressure as they may be expecting.

Pipe X has an angle of Y degrees, but the Wasp-Durand model does not account for inclined or vertical pipes

A [Warning message](#) in AFT Fathom
([SSL Module Only](#))

Full Message Text

Pipe X has an angle of Y degrees, but the Wasp-Durand model does not account for inclined or vertical pipes. Results may be incorrect.

Explanation

This message can only appear when using the [Wasp-Durand Heterogenous Slurry](#) Model.

This message appears when the slope of a pipe is greater than 18 degrees. Pipes with angles between 8 and 18 degrees will display [this caution message instead](#). The Wasp-Durand model does not incorporate [pipe slope corrections](#). [Pipe Slope](#) can impact pressure gradients and deposition velocity. [Vertical conveyance effects](#) will not be captured in the model.

Required Action

If a system contains vertical or significantly-angled pipes, consider using the [detailed Wilson, Addie, Cliff](#) or [4-Component](#) slurry models. In some cases, it may be possible to define the system boundaries to exclude inclined pipes, modeling only the horizontal elements of the piping system.

Pipe X temperature is equal to or greater/less than the maximum/minimum temperature limit of...

A [Warning message](#) in AFT Fathom

There are two variations of this Warning that are similar:

- The following pipes had temperature **equal to or less than the minimum** temperature limit of X set in the Temperature Solver Limits panel under Steady Solution Control in the Analysis Setup window. Results are not reliable.
- The following pipes had temperature **equal to or greater than the maximum** temperature limit of X set in the Temperature Solver Limits panel under Steady Solution Control in the Analysis Setup window. Results are not reliable.

The user has specified limits in the [Temperature Solver Limits panel](#) of the [Steady Solution Control](#) in Analysis Setup (only available when Heat Transfer With Energy Balance is enabled in Fluid Properties). These limits are intended to aid in convergence by restricting the search space while solving, and should

generally be left as the AFT default values. This warning indicates that the final solution had one or more pipes outside the limit range, and the temperature results have been artificially limited by the Solver. The results are therefore not valid. To resolve this either remove or change the temperature limit.

Note: These limits should not be used as Design Alerts, or to approximate some physical phenomenon.

PRV could not control pressure at Junction X

A [Warning message](#) in AFT Fathom

This warning alerts you that the [Pressure Reducing Valve](#) at junction X was not able to control pressure, and backflow may have occurred.

PSV could not control pressure at Junction X

A [Warning message](#) in AFT Fathom

This warning alerts you that the [Pressure Sustaining Valve](#) at junction X was not able to control pressure, and backflow may have occurred.

Pump X had efficiency greater than 100%

A [Warning message](#) in AFT Fathom

Full Message Text

Steady-State:

Pump X had efficiency greater than 100%. This may be due to a density sufficiently greater than the basis of the power input data or an error in the power input data.

Transient with the [XTS Module](#):

Pump X had efficiency greater than 100%. This may be due to a density sufficiently greater than the basis of the power input data or an error in the power input data. First time when efficiency exceeded 100% was (Y seconds)

Explanation

The listed pump had an efficiency over 100% which is mathematically possible, but not physically realistic.

Required Action

Check that the efficiency/power data is entered correctly in the [Pump Configuration](#) window, and that the correct selection is made for either efficiency or power in the dropdown header of the column. If power

data is being used, a reference density may need to be entered to correct the data, see the [Reference Density](#) topic for more information.

If the Variable Speed option is used on the pump, check the control setpoint as it may not be achievable for the given system configuration.

Static pressure lower than vapor pressure at inlet/outlet of pipe/junction X

A [Warning message](#) in AFT Fathom

This warning alerts the users that the static pressure is lower than the vapor pressure in the specified pipe or junction and that cavitation may occur.

Tank Junction X began overflowing at Y seconds

A [Warning message](#) in AFT Fathom

The total volume of fluid entering the tank junction has exceeded the volume of the tank.

Related Examples

Housing Project - XTS1072

Tank Junction X completely drained at Y seconds

A [Warning message](#) in AFT Fathom

The [Finite Tank](#) junction has run out of fluid to supply to the system. The simulation will continue under the assumption that fluid is still being supplied at the inlet pipe. The results past the specified time should only be used to troubleshoot the system as in reality the pipe would pull in the vapor is currently residing in the tank.

Temperature at inlet/exit of pipe X is above/below max/min fluid temperature range

A [Warning message](#) in AFT Fathom

Full Message Text

There are several variations of this Warning:

- Temperature at **inlet** of pipe X is **below min** fluid temperature range.
- Temperature at **exit** of pipe X is **below min** fluid temperature range.
- Temperature at **inlet** of pipe X is **above max** fluid temperature range.
- Temperature at **exit** of pipe X is **above max** fluid temperature range.

Explanation

The temperature used to calculate fluid properties is being predicted outside the valid range of data for the [AFT Standard fluid or User Library Fluid](#). In order to produce results the software is extrapolating data or using a curve fit beyond the defined range of acceptable values. The fluid properties may be incorrect and may affect the result accuracy. The max and min temperature for the fluid data can be seen by opening the Library menu, selecting Edit Fluids, selecting the fluid used in the model, and looking at the top of the Properties section.

Required Action

Adjust the model such that the temperatures remain within the bounds of fluid data. This could take on many forms depending on the goal of the analysis and what is acceptable:

- Check for typos or incorrect inputs in the Model Data (including units), especially at the boundary condition junctions or any components with significant temperature changes
- Increase or decrease the upstream boundary condition temperature (such as the temperature in an Assigned Pressure junction)
- Adjust the heat transfer in pipes such that the temperature does not drop below or rise above the bounds of data
- Add more data to the custom fluid that extends the range to cover the temperatures seen in the model
- Run a scenario using a different [fluid library](#) if the fluid can be found in the other Library sources, and compare the results

Temperature is above the saturation line for the following...

A [Warning message](#) in AFT Fathom

The solution indicates that the temperature and pressure conditions at the specified pipe are such that one or more gas components has reached saturation conditions. This warning only occurs with [Chempak](#) fluids.

Unique solution not possible - Relief valve cycles

A [Warning message](#) in AFT Fathom

The [relief valve](#) cycles between its open and closed state. When closed, the pressure exceeds its cracking pressure and causes it to open. When open, the pressure drop resulting from flow causes the pressure to fall below the cracking pressure and so it closes. When the relief valve changes states, the steady-state Solver must re-run the simulation, but since the relief valve cycles between open and closed each time, a unique solution is not possible. Double check the inputs for the relief valve, the model boundary conditions, and other junction input data.

Velocity in pipe X is less than the deposition velocity

A [Warning message](#) in AFT Fathom

([SSL Module Only](#))

Full Message Text

Steady-State:

Velocity in pipe W is less than the deposition velocity. Results may be inaccurate. (Velocity: Y, Vsm: Z)

Transient with the [XTS Module](#):

Velocity in pipe W is less than the deposition velocity at time X. Results may be inaccurate past this time step. (Velocity: Y, Vsm: Z)

Explanation

Engineers often want to avoid operating [settling slurry systems](#) in such a way that the bulk slurry velocity (V_m) is less than the [deposition/settling velocity](#) (V_{sm}). This warning indicates that the velocity is in fact less than the deposition velocity which can cause operational problems including stationary beds or plugging of pipes. The methods available in the SSL Module cannot model stationary beds, and thus the results may not be accurate under these conditions. The farther the velocity drops below V_{sm} the more uncertainty in the results.

Required Action

Adjust the model such that the velocity is higher than the deposition velocity. If this is not possible, then it may indicate operational problems in the real system, or you must accept that the results will be incorrect.

Potential adjustments:

- Decrease the pipe size
- Increase the flowrate
- Reduce the slope of the pipe

Cautions

Something may be wrong with the model or of interest for the user to double check. Depending on the application, the results may or may not be affected. If results are affected, it would be to a lower degree compared to a Warning. While some Caution messages may be left unresolved without affecting model results, they should never be ignored by the user.

CAUTIONS

Check Valve closed at Junction X

A [Caution message](#) in AFT Fathom

This is a cautionary notice that a [check valve](#) closed, and the Solver will automatically re-run the model.

Default control valve actions at Junction X used when setpoint not achievable because analysis is transient

A [Caution message](#) in AFT Fathom

(XTS Module Only)

The Always Control (Never Fail) option at a [Control Valve](#) was enabled by the user but not used in the solution. The Control Valve [could not control to its setpoint](#) and so default actions were used.

Flow is negative through Junction X. Loss factors may be in error.

A [Caution message](#) in AFT Fathom

This caution alerts you to flow in junctions which is in the opposite direction from the connecting pipes' reference directions. Any [loss information](#) you specified for junctions may be in error because it was defined assuming flow in the other direction.

Flow solution for Junction X is above/below the maximum/minimum value specified for the resistance curve. Flow rate: Y

A [Caution message](#) in AFT Fathom

There are two variations of this Caution that are similar:

- Flow solution for Junction X (Junction Type) is **above the maximum** value specified for the resistance curve. Flow Rate: Y.
- Flow solution for Junction X (Junction Type) is **below the minimum** value specified for the resistance curve. Flow Rate: Y.

This caution alerts you that the calculated flow through junction X is outside the range of the raw data that was entered in the [Resistance Curve](#). The Solver will use the [Curve Fit](#) of the entered data for all calculations, including pressure losses at flowrates that are extrapolated beyond the original data.

The user should check if the curve fit is still a good approximation of the data below the minimum data point, or above the maximum data point.

It is good practice to have the Resistance Curve pass through the point (0,0) since zero flow will result in zero pressure loss. For more information, see the help topic on [Troubleshooting Resistance Curves](#).

Head loss, head rise, hydraulic gradeline and energy gradeline output does not have meaning for zero g systems

A [Caution message](#) in AFT Fathom

If the user sets the gravitational acceleration to zero, presumably for a space application, then the concept of head and gradeline lose meaning. This can be set in [Analysis Setup](#) on the [Environmental Properties panel](#). For example see Equation 1 in [Network Solution Methodology](#) and see that a zero g will lead to a meaningless *HGL*.

Users should ignore all head loss, head rise, and gradelines displayed in the output and focus on pressure results.

Head rise across fixed flow pumps is referenced to system g level...

A [Caution message](#) in AFT Fathom

Full message is:

"Head rise across fixed flow pumps is referenced to system g level. To obtain standard head rise, divide the displayed head rise by the g level."

If the user sets the gravity to something other than standard earth gravity in [Analysis Setup](#) on the [Environmental Properties panel](#), then the head rise at a pump is referenced to that g level and not 1 g.

Junction X has a density change across it

A [Caution message](#) in AFT Fathom

Full Message Text

Junction X has a density change across it. If this is due to a moving interface and not heat input or output then a Volume Balance junction is recommended.

Explanation

This warning is shown when the user has selected Variable Fluid properties on the [Heat Transfer/Variable Fluids panel](#), and the density of the fluid is changing between pipes. The Solver performs a mass balance at junctions, and fluids of different densities on either side of the junction may violate the mass conservation at a junction.

Required Action

If the change in density is due to a change in temperature across the junction, the caution message can likely be ignored. A density change at a junction due to temperature change means the real system likely still has a mass balance at the junction.

If the change in density is due to a change in fluid type or moving interface, the caution message likely should not be ignored. An example of a change in fluid or moving interface is a pipeline carrying fluids of different properties such as oils of different density, or a system is being flushed with water. In this example, the volumetric flowrate is likely conserved across a junction in the real system even if mass is not conserved. The [Volume Balance](#) junction can be used at the fluid interface to capture this situation. However, the Volume Balance is limited in that it cannot capture the interface location changing with time, and is only applicable to steady-state.

Losses at one or more pipes or junctions may be significantly impacted (>25%) by laminar or non-Newtonian conditions, but no correction was made

A [Caution message](#) in AFT Fathom

Full Message Text

Losses at one or more pipes or junctions may be **significantly impacted (>25%) by laminar or non-Newtonian conditions, but no correction was made**. Checking the 'Laminar/Non-Newtonian Correction Strength' output parameter is recommended. If the loss model source was from testing under turbulent conditions, applying a correction is recommended.

Explanation

Flow through one or more pipes or junctions was found to be in the laminar flow regime (Reynolds Number < 2300), but the "Apply Corrections" setting is disabled on the [Laminar and Non-Newtonian Corrections panel](#) in Analysis Setup. The "Laminar and Non-Newtonian Correction Strength" output parameter under either the Pipes or Junctions sections of [Output Control](#) was calculated to be larger than 1.25 for one or more pipes or junctions. This means that if corrections were applied, the resulting equivalent K factors and pressure losses would be significantly modified (>25%) compared to if no corrections were made.

Required Action

If the loss information used in the model was based on manufacturer data that came from turbulent tests, then the user should review the documentation for the [Laminar and Non-Newtonian Corrections panel](#) in Analysis Setup, consider enabling the setting for "Apply Corrections", and review the settings for which the corrections apply. Two scenarios can be compared, one with corrections and one without, to help determine the exact magnitude and location of the impacts. It is also recommended to review the documentation for the [ATKF Method](#).

There is ambiguity in literature about if laminar corrections are required for fittings. A [comparison of literature sources](#) is provided, and it is up to the user to decide when/how to apply the corrections.

If the loss information used in the model was based on true laminar data, then the caution message can be disregarded.

Note: No scenarios have been set in the Multi-Scenario Output display

A [Caution message](#) in AFT Fathom

This caution is when multi-scenario output has been selected in [Output Control](#) but no [scenarios](#) have been selected. As a result, no output can be shown.

Pipe X elevation of Y is below Reservoir Z tank bottom elevation

A [Caution message](#) in AFT Fathom

([XTS Module Only](#))

Full message is:

"Pipe X elevation of Y is below Reservoir Z tank bottom elevation of A."

This is a warning that only occurs when running the [XTS Module](#) when using a finite tank option at a [reservoir junction](#). If the user defines a pipe elevation which is below the bottom of the tank this warning will display.

Pipe X has an angle of Y degrees, but the Wasp-Durand model does not account for inclined or vertical pipes

A [Caution message](#) in AFT Fathom

([SSL Module Only](#))

Full Message Text

Pipe X has an angle of Y degrees, but the Wasp-Durand model does not account for inclined or vertical pipes. Results may be incorrect.

Explanation

This message can only appear when using the [Wasp-Durand Heterogenous Slurry](#) Model.

This message appears when the slope of a pipe is greater than 8 degrees and less than 18 degrees. Pipes with angles above 18 degrees will display [this warning message](#) instead. The Wasp-Durand model does not incorporate [pipe slope corrections](#). [Pipe Slope](#) can impact pressure gradients and deposition velocity.

Required Action

If the slopes of all pipes in the system are below 18 degrees, Wasp-Durand may still be used, but will reflect the system less accurately. Exercise caution if results for uphill pipes show settling maximum velocities near their mean slurry velocities.

Pipe X length is shorter than the elevation change

A [Caution message](#) in AFT Fathom

This caution alerts you to an inconsistency with the input data due to a pipe being defined with a length that is shorter than the elevation change between its connecting junctions. This may be an input error or a result of a junction which itself has an elevation change. If it is due to the junction having an elevation change, you can use different [inlet and outlet elevations](#) at the junction to resolve this.

In this situation, pressure loss along the pipe due to friction is calculated using the pipe length the user has defined while static head loss is determined using the elevation change the user has defined.

Pipe X segment length Y is shorter than the elevation change

A [Caution message](#) in AFT Fathom

The pipe has [intermediate elevations](#) and the length along one of them is less than the elevation change of that segment. In this situation, pressure loss along the pipe due to friction is calculated using the pipe length the user has defined while static head loss is determined using the elevation change the user has defined. This can be corrected by adjusting the intermediate elevations input.

Pipe X steady state Reynold's Number is less than 100. Minor Losses may be significantly lower than expected.

A [Caution message](#) in AFT Fathom

When Reynolds Numbers fall to extremely low values (i.e. less than 100) the fluid enters a flow regime in which the system losses behave in an unintuitive fashion. Based on relationships depicted by the moody diagram, as the Reynolds Number decreases when below 2000, the friction factor and resulting pressure loss increases. In practice, this relationship breaks down and system losses will begin to decrease somewhere between a Reynolds Number of 10 and 100. Due to limited academic knowledge on this subject, and since the Solver is not currently able to model this regime, a caution message is given.

Pump X has no efficiency curve data. Nominal efficiency was used in viscosity correction calculations

A [Caution message](#) in AFT Fathom

This message displays for [centrifugal pumps](#) which use [viscosity corrections](#) to the performance curves. Typical corrections depend on efficiency data which typically depends on flowrate. No such data was provided by the user. However, the user did provide nominal efficiency and that was used in the viscosity correction calculations. Viscosity correction results are less certain in this case.

Pump X has no efficiency curve or nominal efficiency data...

A [Caution message](#) in AFT Fathom

Full message is:

"Pump X has no efficiency curve or nominal efficiency data. An efficiency of 80% was assumed in viscosity correction calculations."

This message displays for [centrifugal pumps](#) which use [viscosity corrections](#) to the performance curves. Typical corrections depend on efficiency data which typically depends on flowrate. No such data was provided by the user. Further, the user did not provide nominal efficiency either. In order to perform the viscosity correction an efficiency of 80% was assumed. Viscosity correction results are less certain in this case.

Pump X properties were outside of the valid range for the viscosity correction calculations. Results may be invalid.

A [Caution message](#) in AFT Fathom

This message displays for [centrifugal pumps](#) which use [viscosity corrections](#) to the performance curves. The methodology used by the [Hydraulic Institute](#) has limits which depend on flowrate, head rise, viscosity, and RPM. If the operating case in AFT Fathom is outside the valid range this warning displays.

Spray Discharge Junction X closed because inflow occurred and Special Condition - No Inflow was set

A [Caution message](#) in AFT Fathom

This is a cautionary notice that a Spray Discharge junction closed because the Solver determined that flow would enter the system, but the No Inflow [Special Condition](#) was set on the Optional tab of the [Spray Discharge Properties window](#). Therefore, the Spray behaved like a [Check Valve](#) and prevented the flow from entering the system.

System Life specified in Cost Settings is different from the XTS time

A [Caution message](#) in AFT Fathom

([XTS Module Only](#))

The System Life as specified in [Cost Settings](#) is of a different length than the transient duration specified in the [Simulation Duration panel](#).

The following Detailed Tee/Wye junctions have a different automatic orientation than defined on the Workspace

A [Caution message](#) in AFT Fathom

Full Message Text

The following Detailed Tee/Wye junctions have a different automatic orientation than defined on the Workspace:

Tee/Wye: J1, J2, J3, J4, J5, J6, J7, J8, J9, J10 and X more junctions. See the Model Diagnostics window for more information.

Explanation

The listed [Tee/Wye junctions](#) are using the [Detailed Loss Model](#) with the Arrangement Type set to "Auto", however, the automatic pipe orientation saved for the Tee/Wye Properties window is not consistent with the current orientation on the Workspace. For the K factors to be calculated correctly, the connections for pipes A, B, and C must be defined correctly. For example, on a 90 degree Tee, the branching flow pipe (C) in the Tee/Wye Properties window must match the intended branching pipe on the Workspace. If the branching pipe C was switched with the straight run pipe B, then the K factors would be switched. This can give incorrect results which may propagate through the model and compound with other Tee/Wye junctions since the Detailed Loss Model is dependent on flows and pressures.

A full list of the affected junctions can be found on the [Model Diagnostics window](#) (Tools menu > Model Diagnostics > Junctions > Tee/Wye > "Tee orientation inconsistent to workspace?").

Required Action

First, it is recommended to align the pipes on the Workspace in a way that resembles the true geometric orientation of the junction as depicted by either the [2D or Isometric grid](#). If there is a 90 degree angle, draw the pipes in a way that depicts this in the proper way. Furthermore, point the pipe [Reference Positive Flow Direction](#) towards the expected flow direction. The Auto Arrangement Type will utilize the directions and angles between each pipe when attempting to automatically configure the pipe connections. Therefore, following these modeling practices can increase the success rate of properly identifying and applying the K factors to the correct pipes. The tips below can be used to help quickly apply these principles:

- The pipe direction can be switched by using the Reverse Pipe Direction icon on the Workspace Toolbar or using the F3 keyboard shortcut.
- Display the Workspace Grid and enable Snap to Grid from the Arrange menu.
- Rotate the Tee/Wye junction icon such that it is consistent with the pipes by right-clicking the icon and selecting either "Auto-rotate Icon" or "Customize Icon".

Next, open each Tee/Wye Properties window listed in the Caution message or in the Model Diagnostics window, configure and verify the proper pipe connections, click OK, and re-run the model. If needed, the Arrangement Type can be switched to "User" to guarantee that the junction retains the proper configuration.

If the losses due to the Tees/Wyes are negligible compared to the total system pressure drop, the user can consider switching to the Simple Loss Model which assumes the junction is lossless. This would be equivalent to morphing the junction into a [Branch](#).

The performance of Pump Junction X may be affected by the fluid viscosity

A [Caution message](#) in AFT Fathom

Full message is:

"The performance of Pump Junction X may be affected by the fluid viscosity. Apply the Viscosity Correction on the Pump Property window and rerun the model to assess the impact of the fluid viscosity."

This caution is shown if the fluid used in the model is not water, and the pump viscosity correction factors would be less than one based on ANSI/HI 9.6.7-2015. If the pump and fluid used in the model is in the scope of the ANSI standard it may improve accuracy to use viscosity corrections at the pump.

See the [Pump Viscosity Correction](#) topic for more information.

Transient data did not extend to Stop Time at Junction X

A [Caution message](#) in AFT Fathom

[\(XTS Module Only\)](#)

The transient data at Junction X does not have time data that covers the entire time frame of the simulation as defined on the [Simulation Duration panel](#). The final data point will be used for the remaining time steps of the simulation. You can enter additional time data points at Junction X to avoid this message.

Wasp-Durand relative concentration (X) exceeds 0.8

A [Caution message](#) in AFT Fathom

Full Message Text

Wasp-Durand relative concentration (X) exceeds 0.8 and relative concentration for d95 particles (Y) exceeds 0.5 in pipe Z, indicating the slurry only consists of a pseudo-homogeneous component. However, the Liu Dezhong Model accounts for both pseudo-homogeneous and heterogeneous components.

Explanation

In quasi-homogeneous flow conditions, there are few solids in the heterogeneous slurry component with nearly all contributing to the homogeneous component. The heterogeneous gradient has little contribution to the overall pressure and flow results, but [Wasp-Durand](#) methods may still be appropriate for modeling the pseudo-homogeneous component.

Required Action

As the pseudo-homogeneous component is dictating results almost exclusively, modeling the homogeneous hydraulics accurately is more important than capturing heterogeneous effects in these pipes. Modeling the apparent viscosity appropriately is therefore of particular importance under these conditions.

If the entire system is quasi-homogeneous and the slurry is expected to exhibit [non-Newtonian](#) behavior, it may be better to model the system as a non-Newtonian fluid rather than as a settling slurry.

However, if rheological data is unavailable, use of the Wasp-Durand slurry model with Fei Xiangjun or Thomas viscosity models is likely to provide more accurate results than modeling the fluid as a pure Newtonian fluid.

Information

An Information message is intended to make the user aware that an advanced feature is being used in the model, or to provide comments or suggestions relating to an advanced feature. Following suggestions may achieve higher accuracy results if the right conditions apply, but can also cause incorrect results if not applied properly. If the user understands the feature then no action is required.

INFORMATION

Steady-state conditions are laminar in the following pipes

An [Information message](#) in AFT Fathom

Full Message Text

There are several variations of this Information message:

- Steady-state conditions are laminar ($100 \leq Re \leq 2300$) in the following pipes. Loss factors are typically from turbulent tests and may not be accurate for connected junctions or fittings in these pipes. Laminar and Non-Newtonian Corrections **are not applied**.
Pipes: P1, P2, P3, ...
- Steady-state conditions are laminar ($100 \leq Re \leq 2300$) in the following pipes. Loss factors are typically from turbulent tests and may not be accurate for connected junctions or fittings in these pipes. Laminar and Non-Newtonian Corrections **are applied with default settings**.
Pipes: P1, P2, P3, ...
- Steady-state conditions are laminar ($100 \leq Re \leq 2300$) in the following pipes. Loss factors are typically from turbulent tests and may not be accurate for connected junctions or fittings in these

pipes. Laminar and Non-Newtonian Corrections **are applied with user-modified settings**.

Pipes: P1, P2, P3, ...

Note: If a [Zero-Length Connector \(ZLC\)](#) had laminar flow, there will be an additional note appended to the message stating that ZLCs have been excluded from the list of pipes. ZLCs cannot have Fittings & Losses, so the message does not apply to them.

Explanation

This Information message is ultimately intended to communicate that pressure losses in junctions and fittings may be different under laminar flow conditions compared to turbulent conditions. If flow through a junction or fitting is laminar, then extra consideration may be warranted for how this will impact the pressure drop, and if corrections should be made. Fathom offers methods to correct losses for laminar flow, but there is also [debate in literature](#) about if corrections should be made. Specifics will be discussed below, otherwise consult the Required Action section below.

It should be noted that this message is not intended to discuss pressure drop from pipe friction. Some notes on laminar pipe friction will be provided at the end of this explanation.

Turbulent vs Laminar Loss Coefficients

It is generally believed that the pressure drop through junctions and fittings derive from the physics of bulk fluid motion, wall separation, turbulence, etc. Therefore, the losses during laminar flow may be different compared to turbulent conditions. These pressure losses are typically modeled using data from manufacturers ([Resistance Curves](#) and [Valve Cv or Kv](#) values) or loss coefficients from academic references ([K Factors](#)). The problem is that data from these sources are typically based on laboratory tests which usually occurred under turbulent flow conditions with water.

Laminar flow is often associated with high viscosity fluids, and usually the pressure drops through junctions are higher under laminar conditions. For example, a manufacturer may provide a pressure loss data point at a specified flowrate for a strainer that was tested with turbulent water in their laboratory. If this strainer is installed in a system that pumps high viscosity oil, it is very likely that the pressure drop through the strainer at the same specified flowrate will be much larger. Therefore, it is important to consider these effects when designing systems to ensure they can meet design requirements.

Correction Options

The [Laminar and Non-Newtonian Corrections panel](#) in Analysis Setup offers some capabilities that can help estimate adjusted loss coefficient values for laminar flow. AFT generally recommends to compare scenarios with and without the [Adjusted Turbulent K Factor Method \(ATKF\)](#) enabled to determine the magnitude of effect. Results can be easily compared using [Multi-Scenario Output](#).

By default the laminar flow regime is defined by a Reynolds Number less than 2300, but this value can be changed on the [Environmental Properties panel](#) of Analysis Setup. The lower bound of $Re = 100$ exists because there is a [separate Caution message](#) if the flow is in the "deep laminar" flow regime which is defined by Reynolds Numbers less than 100. The Reynolds Number for a pipe can be checked by adding the "Reynolds Number" output parameter from the Pipes section of the [Output Control](#). The "Laminar and Non-Newtonian Correction Strength" output parameter can be added from both the Pipes section and Junctions section of the Output Control. The correction strength parameter for pipes will refer to the K Factors for any lumped Fittings & Losses, and not the pipe friction itself. A correction strength larger

than 1.0 acts as a multiplier that increases the equivalent K Factor and pressure loss for the pipe fittings or junction.

Comment on Area Restrictions Inside Junctions

When flow in a pipe is laminar, there is also an implication that flow through connected junctions would also be laminar. This also applies to components and fittings that have been lumped into the pipe [Fittings & Losses](#). One caveat to mention is that in reality some junctions could have an area restriction such as the path of flow through a valve, orifice, etc. In an area restriction, the velocity will increase, potentially to the point where the Reynolds Number is no longer laminar, and laminar corrections might not be needed. This might be a consideration for systems that operate in the transitional flow regime. Even when accounting for area restrictions within junctions, there is a high probability that if the connected pipe experiences laminar flow that the junction will also experience laminar flow, or at least operate under a condition with different pressure loss characteristics compared to reference K Factors.

Pipe Friction Under Laminar & Non-Newtonian Conditions

For more information on how to account for laminar or Non-Newtonian effects on pipe friction, consult the [Pressure Drop in Pipes Detailed Discussion](#) and [Viscosity Model panel](#) topics. Below is a brief summary:

- If the pipe uses the default Friction Model (Standard), then the Solver will use the Darcy-Weisbach loss model which has an explicit correlation for the laminar friction factor ($f=64/Re$). In this case laminar flow is automatically accounted for in pressure drop calculations. Other Friction Models may or may not account for laminar flow - consult the equations for each friction model on the Pressure Drop in Pipes Detailed Discussion topic.
- If a Non-Newtonian viscosity model is used, then the friction factor will be calculated according to the corresponding methods - consult the equations for each viscosity model.

Required Action

First, the user should consult the help topic on the [debate of if laminar corrections are required](#) for junctions and fittings, and use their engineering judgment to decide if they believe corrections are required. If the user does not want to apply corrections, then keep the laminar corrections disabled on the [Laminar and Non-Newtonian Corrections panel](#) (this is the default selection).

If corrections are desired, and if junction loss coefficients were from turbulent conditions, then it is recommended to enable laminar corrections. The user should consider enabling the [Adjusted Turbulent K Factor method](#) and read the corresponding help topic to learn more.

Note: All of the K Factors from Fathom's internal [Fittings & Losses Library](#) are from sources that tested under turbulent flow conditions.

If loss factors were from laminar conditions and this is confirmed with the manufacturer or reference source, then keep the laminar corrections disabled so as to not double count the effect of laminar conditions. Corrections can be enabled or disabled for an entire junction type. This may be useful if you are accounting for laminar flow in bends and valves using the [3-K Darby Method](#), but not for General Component junctions, and therefore you only want to apply corrections to General Component K Factors.

Previous Variations of the Messages

From the initial release of Fathom 13 to an intermediate maintenance release there were alternate variations of these messages (release 2023.09.14 up to and including 2024.02.05). Changes were made because AFT changed our philosophy on how and when to alert users. Previously an Information message would communicate when significant corrections were made (>25%), and a Caution message would communicate if no corrections were made but would have been significant if applied. Whether the Information or Caution message was displayed depended on if the user enabled corrections from the [Laminar and Non-Newtonian Corrections panel](#). Models created in Fathom 12 or previous versions had laminar corrections enabled by default, and importing them into Fathom 13 will keep the corrections enabled. New models created in Fathom 13 have laminar corrections disabled by default, but the Informational message documented on this topic is shown to notify the user. The new philosophy is to simply alert users to laminar flow, which may or may not be expected, and allow the user to decide if applying corrections is required.

Previous Information message:

Losses at one or more pipes or junctions were **significantly modified (>25%) with a Laminar or Non-Newtonian correction**. The strength of the correction can be seen by checking the Laminar/Non-Newtonian Correction Strength output parameter. If the loss model source was from testing under laminar conditions, applying a correction is NOT recommended.

Previous Caution message:

Losses at one or more pipes or junctions may be **significantly impacted (>25%) by laminar or non-Newtonian conditions, but no correction was made**. Checking the 'Laminar/Non-Newtonian Correction Strength' output parameter is recommended. If the loss model source was from testing under turbulent conditions, applying a correction is recommended.

Troubleshooting

This section discusses methods of troubleshooting models that are having difficulty converging. Topics include discussions of common issues, ideas to simplify overly complex models, and methods for adapting the solution methodology through either tolerance or flow relaxation.

Contents

Diagnostics Window	797
Use the Model Data Window	798
Transfer Results to Initial Guesses	800
Poor Pump Curve Fits	800
Troubleshooting Resistance Curves	801
Use the Output Window Sort Feature	802
Tee/Wye Junction Complexity	803
Lower the Flow Rate Relaxation	803
Try Absolute Tolerance	804
Make Initial Flow Rate Guesses for Pipes	804
Review the Iteration History	804
Change Boundary Conditions	805
Turn Off Parts of the Model	805
Break a Large Model Into Submodels	806
Identifying and Restoring Corrupted Model Files	807

Diagnostics Window

The Model Diagnostics window is a feature introduced in AFT Fathom 11. It is intended to help users check on a number of settings that may have been changed during the model-building process and that may affect the model results.

The Model Diagnostics window can be accessed from the [Tools menu](#).

When the Model Diagnostics window is opened all input will be checked, and the results will be displayed in a tabular format. Items of note will be highlighted based on the impact each identified item may have on the results. It should be noted that **not all items identified by this window will have a negative impact on results**. Rather, the identified items are settings that have frequently been incorrectly applied by other users. This is not an exhaustive list of troubleshooting items, though the most common issues are all included.

Data in the Model Diagnostics is organized by the Data Source, which is the window where that setting/input is defined. Any item that may negatively impact the model will be highlighted based on its potential to affect the model, as shown in the Potential Impact column. For pipe and junction diagnostic items, the pipe(s)/junction(s) that were identified will be listed below the diagnostic item. For an example, see Figure 1 below. The Diagnostic window indicates that pipe P1 has unusual pipe dimensions, and should be reviewed.

Data Source	Diagnostic Description	Result	Potential Impact	Diagnostic Details
Fluid Properties	Fluid with unusual property values?	False	None	A fluid property is outside the typical range. (Density 30 - 100 lbm/ft3) (Viscosity 0.5 - 250 lbm/hr-ft) (Vapor Pressure > 1000 lbm/ft2)
Fluid	Non-Newtonian Viscosity Model?	True	Low	The fluid Viscosity Model is anything other than Newtonian.
Heat Transfer/Variable Fluids	Non-constant fluid properties?	False	None	Fluid Properties is set to Variable or a Heat Transfer mode.
Laminar and Non-Newtonian Corrections	Laminar and Non-Newtonian Corrections modified?	False	None	The Laminar and Non-Newtonian Correction method or losses it is applied to has been changed from the defaults.
Steady Solution Control			None	
Environmental Properties			None	
Miscellaneous			Low	
NFPA	NFPA enabled?	True	Low	NFPA has been enabled.
Advanced Parameters	Advanced Parameters Modified?	False	None	Any Advanced Parameter has been modified from the default.
Advanced Parameters	Heat Transfer Sectioning modified?	False	None	Heat exchanger or pipe thermal sectioning has been modified from the default.
Pipes			High	
Pipe Model	Friction model not Absolute?	False	None	The following pipes have a Friction Model other than Absolute Roughness.
Pipe Model	Unusual pipe dimensions?	True	High	The following pipes have an unusual geometry. (Length < 0.5 ft or Diameter < 1" or Diameter > 50% Length)
P1 (Pipe)				
Pipe Model	User specified size?	False	None	The following pipes use a User Specified Material.
Optional	Unusual Design Factors?	False	None	The following pipes have Design Factors less than 0.5 or greater than 5.
Junctions			Unknown	
Output			Unknown	
Steady	Laminar corrections enabled and laminar flow exists?	N/A	Unknown	[No output to test] The following pipes have laminar or transitional non-zero flow and a Laminar Correction has been enabled.
Steady	Negative absolute pressures exist?	N/A	Unknown	[No output to test] The following pipes have steady pressures below zero absolute.
Steady	Power/Bingham Re > 100,000?	N/A	Unknown	[No output to test] The following pipes are using Power Law or Bingham Plastic Non-Newtonian fluid models with a laminar correction.

Figure 1: Example of the Model Diagnostics window

Related Topics

Troubleshooting 797

Use the Model Data Window

Besides its usefulness for documentation purposes, the [Model Data window](#) is a powerful tool for finding input errors in your model.

Look at the model in Figure 1. If you suspected an input error in one of the pipes, the task of looking for the error in each individual pipe input window would be tedious.

Now look at the Model Data window for the pipes in Figure 2. Here it is much easier to scan down columns of data looking for input errors. In this case, one of the pipe lengths was accidentally entered in miles rather than feet.

Troubleshooting

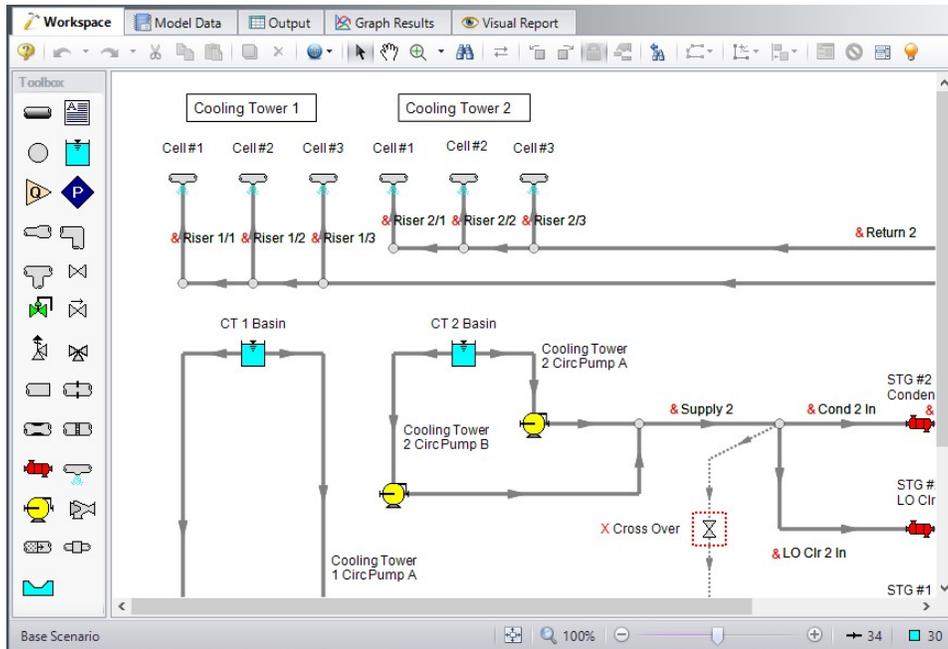


Figure 1: Workspace layout of a pipe system with an input error in one of the pipes (error not shown)

Pipe	Name	Pipe Defined	Length	Length Units	Hydraulic Diameter	Hydraulic Diam. Units	Friction Data Set	Rough
100	Circ Pump 1A Suction	Yes	10 feet		19.25 inches		Standard	
101	Circ Pump 1A Disch	Yes	20 feet		19.25 inches		Standard	
102	Supply 1	Yes	1000 feet		29 inches		Standard	
103	LO Clr 1 In	Yes	20 feet		8.125 inches		Standard	
106	LO Clr 1 Out	Yes	20 feet		8.125 inches		Standard	
107	Cond 1 In	Yes	50 miles		27 inches		Standard	
108	Cond 1 Out	Yes	50 feet		27 inches		Standard	
109	Return 1	Yes	1000 feet		29 inches		Standard	
110	CT 1 Hdr A	Yes	40 feet		23.25 inches		Standard	
111	CT 1 Hdr B	Yes	40 feet		17.376 inches		Standard	

Figure 2: Model Data window display for the model in Figure 1. From this perspective it is much easier to see that one of the pipes (#107) has length data entered in miles rather than feet. This is not automatically an input error, as AFT Fathom accepts pipe input with mixed engineering units. However, more frequently than not the user intends to use consistent units, so this probably is an error.

Related Topics

Troubleshooting	797
-----------------------	-----

Transfer Results to Initial Guesses

AFT Fathom’s iterative Solver (the part of AFT Fathom that applies the governing incompressible flow equations to obtain a solution to the pipe system) takes initial guesses (also called initial conditions) and applies the principles of mass, momentum and energy balance to improve the guesses. This is called iteration. It continues to refine the solutions until agreement with the [governing equations](#) is obtained.

After a converged answer is obtained, you can choose to take these results and use them as the initial guesses for a subsequent run. This causes much faster convergence in subsequent runs because you are starting the solver at or very close to the actual solution.

This feature is called Transfer Results to Initial Guess, and it is located on the Output window [Edit menu](#) and [Toolbar](#).

Related Topics

Optional Input for Junctions	255
Optional Input for Pipes	238
Make Initial Flow Rate Guesses for Pipes	804
Troubleshooting	797

Poor Pump Curve Fits

The [Pump Properties window](#) allows [curve fits](#) of pump data up to fourth order. Depending on the number of data points entered, pump curves fits that look like the orange curve in Figure 1 are possible.

AFT Fathom interprets flow rates past the inflection point around 80 as acceptable operating points, which can lead to unusual results or non-convergence. Users should always ensure that their pump curves cross the flow rate axis as shown in the blue curve in Figure 1.

AFT Fathom will attempt to warn the user when a model containing a pump curve fit as seen in Figure 1 is run. The user can choose to run the model with this curve fit anyways, but they should take care that the pump is not operating to the right of the inflection point.

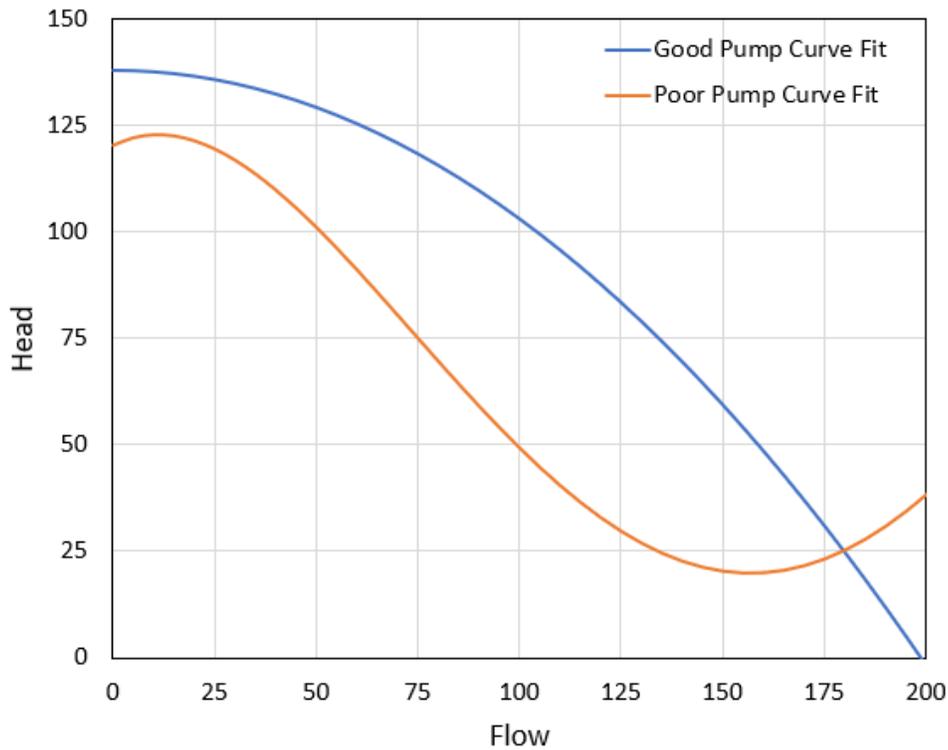


Figure 1: A good pump performance curve fit in blue and a poor pump performance curve fit in orange. All pump curve fits should include sufficient data such that the curve crosses the zero pressure rise axis as shown in the blue curve.

Related Topics

Pump.....	291
Pump Configuration.....	296
Pump Efficiency Data.....	328
Troubleshooting.....	797

Troubleshooting Resistance Curves

Common issues with resistance curves include curves where losses do not increase with flow, and curves with a non-zero loss at zero flow. Figure 1 below shows examples of curves with potential issues compared to a correctly defined curve.

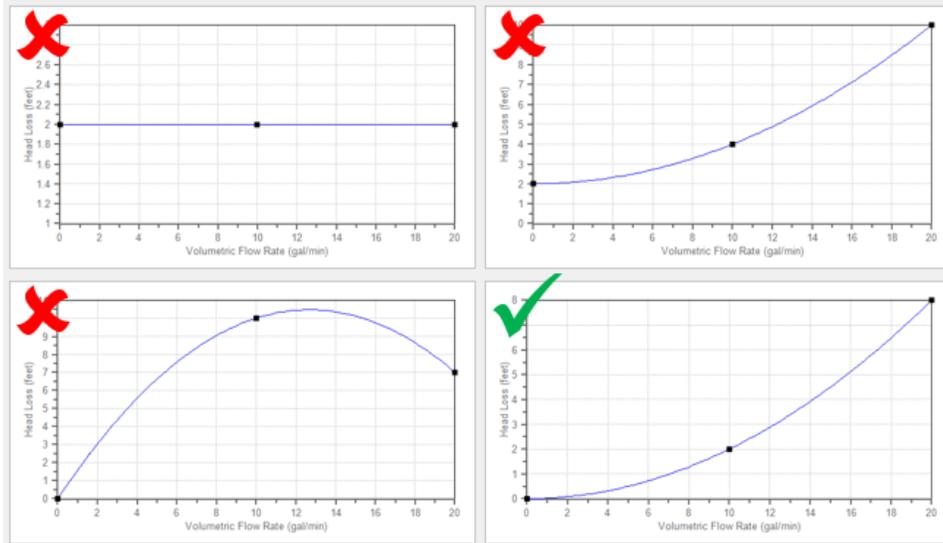


Figure 1: Example resistance curves showing correct and incorrect curve types

These common issues create two potential problems in a model:

1. If the system sees a constant pressure drop even at low flowrates, the iterative Solver can encounter difficulty converging on a solution since the system flow and pressure drop are no longer dependent on each other.
2. Real equipment in a system will typically cause a higher pressure drop at larger flowrates, and will not provide any loss to the system at zero flow. Users should capture a single known pressure loss data point by defining a curve which passes through two points: the known pressure drop at the tested flowrate, and zero pressure drop at zero flow. The [Resistance Curve](#) topic gives more information on defining this type of curve.

Related Topics

Resistance Curve Loss Model.....	264
General Component.....	282
Heat Exchanger.....	282
Troubleshooting.....	797

Use the Output Window Sort Feature

If a model runs but the results look incorrect, use the **Sort feature** found in the [Arrange menu](#) or toolbar in the [Output Window](#) to look for extremes in velocity, pressure drop or other parameters. An input error (like incorrect diameter) may be easier to find by looking at its effect on the results.

Related Topics

Troubleshooting.....	797
----------------------	-----

Tee/Wye Junction Complexity

[Tee/wye junctions](#) contain powerful and sophisticated models for determining the pressure losses at flow splits and converging flow streams. This power comes at the cost of additional model complexity, which can lead to convergence problems in models with numerous tees. Here are a couple of recommendations.

Tee/Wye junctions have two loss options: Simple and Detailed. The Simple loss model ignores losses at the tee. The Detailed model uses the correlations summarized in the [Tee/Wye Loss](#) topic. The potential convergence problems occur when using the Detailed loss model.

The potential problem is not with the Detailed loss model itself. Rather, it is due to multiple tees coupling together in such a way that the Solver cannot converge on all of the flow splits. The tee loss coefficients are dependent on the flow splits, but the flow splits are dependent on the other losses in the system.

Therefore, if a pipe network contains many tees using the Detailed loss model, when one tee's loss coefficient is updated from an intermediate Solver iteration, the entire network needs to be re-calculated to determine how the flow splits changed. Sometimes this process is not able to terminate within the Tolerances specified by the model.

A common issue with detailed tees is having the pipes connected in the Properties window differently than how connections are set up in the Workspace. Make sure to double check those connections before running the models.

You can quickly and easily check if a convergence problem is being caused by using the Detailed loss model. Create a child scenario, then using the [Global Junction Edit window](#), you can change all tees to the Simple loss model and rerun the model. If it now converges then the problem was likely due to the use of Detailed tee modeling.

Experience has shown that this problem can frequently be resolved by using absolute tolerance criteria on the [Tolerance panel](#) of [Analysis Setup](#) rather than relative tolerance. Use this option at your own discretion and avoid setting the tolerances too loose such that you get false convergence. Often times, using the "Both Absolute and Relative" option gives the best of both worlds.

You also have the option of just using the Simple (lossless) tee model for all tees. In many systems the loss due to tees/wyes is negligible, especially for systems which see significant loss from other components that dominate the system.

Related Topics

Tee/Wye.....	341
Tee/Wye Loss Model.....	342
Troubleshooting.....	797

Lower the Flow Rate Relaxation

The [Iterations panel](#) in [Analysis Setup](#) allows you to change the numerical relaxation parameter for both flow rate and pressure. It is recommended that you use only 0.5 or 1.0 for pressure relaxation, but lower values of flow rate relaxation can be very helpful.

A good first step is to lower the flow rate relaxation to 0.05 or 0.1. By lowering the flow rate relaxation you improve AFT Fathom's ability to obtain convergence. The cost is slower run times. In some cases accepting the slower run times that come with flow rate relaxation is the only means of obtaining convergence.

You can lower the relaxation values as low as you want, but it is recommended it not be lowered below 0.01.

Related Topics

Troubleshooting	797
-----------------------	-----

Try Absolute Tolerance

Most models converge more reliably with relative tolerances, which is the default on the [Tolerance panel](#) in [Analysis Setup](#). However, some models converge much better with absolute tolerance. This most often applies to models which have very small values for flow or pressure, as it is much harder to meet a relative tolerance using small numbers.

If a model will not converge with relative tolerance, you can try changing to absolute tolerance. Even better, try the option to converge on either absolute or relative, which will offer the most flexibility.

Related Topics

Troubleshooting	797
-----------------------	-----

Make Initial Flow Rate Guesses for Pipes

If you do not provide any [initial guesses](#), AFT Fathom will make its own guesses. Occasionally the AFT Fathom guesses are far away from reality. For some models providing a crude initial guess for all the pipe flow rates in the system can help the model converge.

The crude guesses can be as simple as selecting all the pipes and initializing all of them to 1 lbm/sec, even though some operate at 0.1 lbm/sec and others at 10 lbm/sec. However, note that initial guesses that are too far from the final solution can also interfere further with convergence.

Related Topics

Optional Input for Junctions	255
Optional Input for Pipes	238
Transfer Results to Initial Guesses	800
Troubleshooting	797

Review the Iteration History

The [Iteration History window](#) can be opened from the [Solution Progress window](#). You can use this to see which pipes or junctions are farthest out of tolerance, which may direct your troubleshooting efforts towards these elements.

Related Topics

Solution Progress Window.....	383
Iterations Panel.....	165
Troubleshooting.....	797

Change Boundary Conditions

Changing the boundary pressures or flow rates can sometimes give you insight into what is giving the model trouble. For example, obtaining convergence only within a certain range of flowrates could indicate an issue with curve fit data being unrealistic outside of that range of flowrates.

Related Topics

Troubleshooting.....	797
----------------------	-----

Turn Off Parts of the Model

Using the [Special Conditions](#) feature, you can turn off elements in your model and in effect shutdown entire sections. If the model runs when certain sections are closed, this could indicate a problem with one of the components in the closed section.

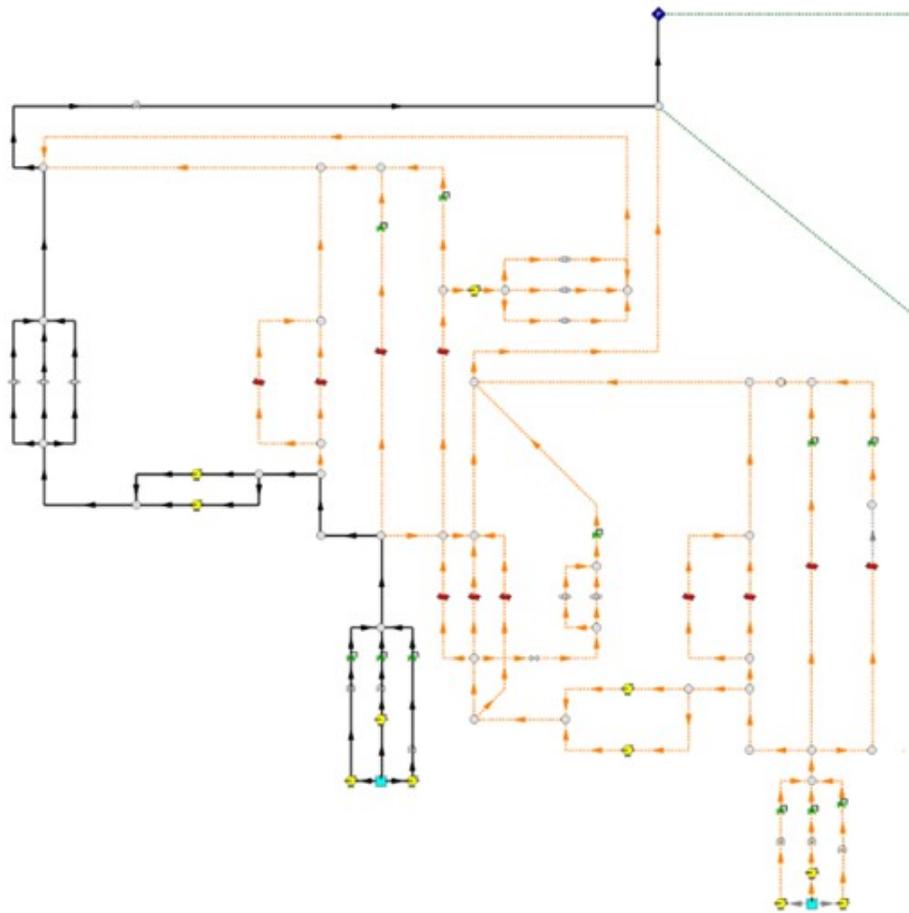


Figure 1: The pipes in black are open to flow, while the pipes in yellow have been closed using special conditions, meaning that Fathom will automatically set their flow solution to 0

Related Topics

Special Conditions	213
Optional Input for Junctions	255
Optional Input for Pipes	238
Troubleshooting	797

Break a Large Model Into Submodels

AFT Fathom provides convenient features for deleting parts of your model and merging parts back in. A section of your model can be quickly deleted by selecting it on the [Workspace](#) and choosing Delete from the [Edit menu](#). If the section is instead Cut, it can be Pasted into a new model of its own for testing or holding. At a later time the removed sections can be added back in to the current model by using the [Merge](#) feature on the File menu.

The advantage of breaking your model into sub-models is that you can investigate the convergence characteristics of each part of your model independently. If you can get each sub-model to converge, you should be able to get the original model to converge also. If one of your sub-models will not converge, it is doubtful your original model will converge either.

Related Topics

Troubleshooting 797

Identifying and Restoring Corrupted Model Files

The most common way to tell if a model file is "corrupted" is an error message that says "An error occurred reading the file into AFT Fathom. Can't find [TOOLBOX PREFERENCES] heading."

You may see multiple of these messages after pressing OK, and the part between the brackets [] may say something else each time. This is indicating that the software is trying to read different sections of the file, and it is finding that there is either no data, or corrupted data. It is then showing you an error for each section of the file that the software tries to read when loading the model, similar to Figure 1.

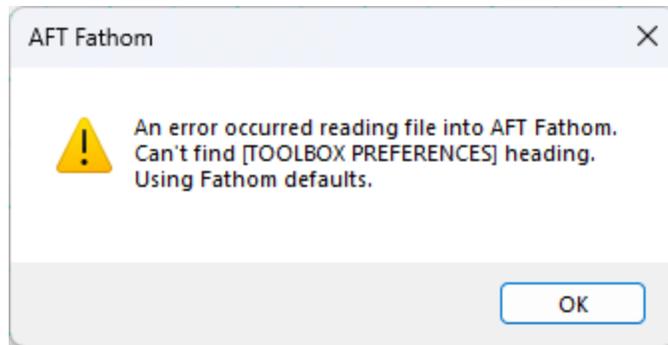


Figure 1: Error message indicating that some portion of the model file cannot be found

Another indicator is if the model's file size has been reduced to 0 or 1 KB. This indicates the data inside the file has been lost. This can be seen in the file size in the Windows File Explorer (Figure 2).



Figure 2: Very low file sizes can be an indication of model file corruption

What are the backup files (.bak extensions)?

AFT software will automatically generate backup versions of your model file each time you save. By default, these files will be located in the same directory where you have saved your model. These files have a specific naming convention, but generally the file extension will start with ".bak" and end with three numbers. Newer versions of AFT software will have a fourth letter after the .bak, and it signifies which application the backup is for.

- AFT Fathom.**bakF-001**
- AFT Arrow.**bakA-001**
- AFT Impulse.**bakI-001**
- AFT xStream.**bakX-001**

The numbers signify previous versions of the model, with the 001 extension being the most recent backup. Of course, if your current model file is corrupted and you need to restore a backup, the 001 backup won't contain the most recent updates you made to the model, but it will be better than losing everything! The backups named 002 and 003 are the second and third most recent backups, respectively. Generally, you won't need to use these files unless you want to revert to even earlier state of the model file, or there is an issue with the 001 backup.

When you save a model file, the application will copy the previous version of the file into .bak001, the previous .bak001 will be copied into .bak002, and so on, creating a cascading effect. This way .bak001 is always the most up to date backup version of the model.

How to Restore a Backup File

1. Close the AFT application if it is still open.
2. In the same location as your model file, locate the file with the extension *.bak-001.
3. Create a copy of *.bak-001. This is useful in case something goes wrong, and you need to try again.
4. Rename the copied file and change the file extension to the appropriate application file extension below. Windows will prompt you to confirm the extension change; click Yes.
 - Fathom: *.**fth**
 - Arrow: *.**aro**
 - Impulse: *.**imp**
 - xStream: *.**xtr**
5. Try opening the restored file. If this didn't solve the problem, you can try the same steps with an earlier backup file, using .bak-002 or .bak-003 if they exist. However, note that doing so will revert even more changes because these are even earlier versions of the model from multiple saves prior.

Name	Date modified	Type	Size
MM_2022-12-29.bakF-001	12/29/2022 5:44 PM	BAKF-001 File	78 KB
MM_2022-12-29.bakF-002	12/29/2022 3:51 PM	BAKF-002 File	88 KB
MM_2022-12-29.bakF-003	12/29/2022 2:11 PM	BAKF-003 File	88 KB
MM_2022-12-29.fth	12/29/2022 6:33 PM	AFT Fathom 12 File	78 KB

Figure 3: The newest backup file, identifiable by file extension numbering and date modified

Name	Date modified	Type	Size
MM_2022-12-29 - Restored.fth	12/29/2022 5:44 PM	BAKF-001 File	78 KB
MM_2022-12-29.bakF-001	12/29/2022 5:44 PM	BAKF-001 File	78 KB
MM_2022-12-29.bakF-002	12/29/2022 3:51 PM	BAKF-002 File	88 KB
MM_2022-12-29.bakF-003	12/29/2022 2:11 PM	BAKF-003 File	88 KB
MM_2022-12-29.fth	12/29/2022 6:33 PM	AFT Fathom 12 File	78 KB

Figure 4: Changing file name extension to restore the backup file

Tips to Prevent File Corruption

The best way to prevent file corruption is to open, edit, and save the file locally on your PC. It is not recommended to edit and save files directly to a shared server, network drive, or cloud storage platform like Dropbox or OneDrive.

If the connection to the network location is interrupted or cuts out while saving the model file, it could corrupt the file or cause data loss. This can be especially common on VPNs. If you need to have the file in a shared location for colleagues to access, it is recommended that you download the file to your local PC, edit/save the file locally on your machine, then upload the file afterwards.

Customize Your User Settings Related to Backups

By default, only a maximum of three backups will be kept at a time. If desired, you can change this number in Number of Backup Files Created, which can be accessed in [User Options](#) as shown in Figure 5. Here, you can also choose to check the box that will automatically save your model in a specified time interval.

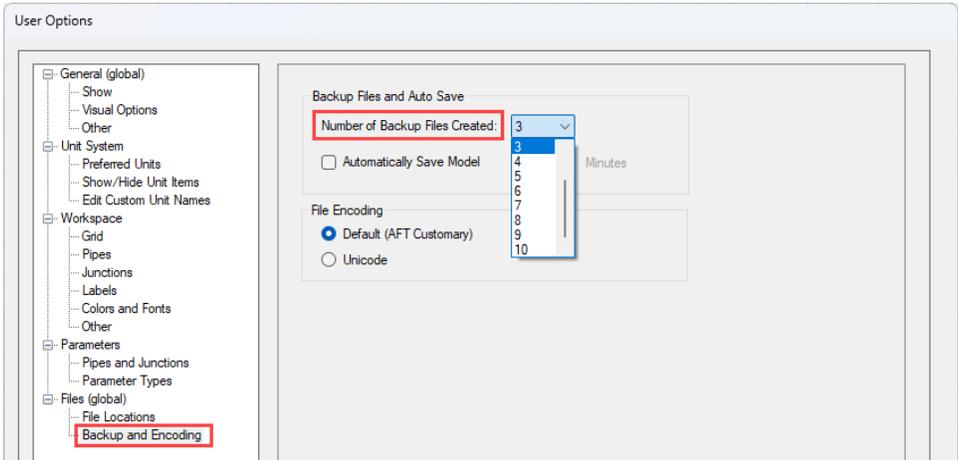


Figure 5: The number of backup files created and options to save automatically can be customized in User Options

Related Topics

Troubleshooting 797

Technical Support

This section focuses on the technical support options for AFT Products. Topics include descriptions of the various support options, contact information for the AFT Support team, and links to external sites to assist with installation issues related to AFT Fathom.

Contents

Technical Support Options	811
Contacting AFT	812
Information on Software Installation	813
Chempak Database Installation	813
Output Files	815
Updated Fluids Library	815
Special File Control Features	816
Converting Models from Earlier Versions	817

Technical Support Options

Support, Upgrade, and Maintenance (SUM)

This primary level of support is for assistance with specific features within the software, or resolving issues with a model such as difficult to converge models. AFT is limited in the ability to interpret model results and cannot recommend specific changes in a system.

One full year of SUM is provided with the purchase of a full license. Leases have SUM coverage during the period of the lease. Further support after the first year requires SUM renewal.

For technical support questions falling in this category, [contact AFT](#).

Flow Expert Package (FEP)

The FEP is a higher level of consulting assistance for a particular project. This service will assign a qualified AFT engineer to assist in not only modeling your system, but also with providing operational suggestions and interpretation of results.

The intention of this service is to provide assistance to active users of AFT software and address questions on the appropriate use of the software for specific system requirements.

Please visit our website at: <https://www.aft.com/products/aft-flow-expert-package> for more information on our FEP services.

Flow Modeling Consulting Services

AFT's sister company, Purple Mountain Technology Group (PMTG), provides comprehensive consulting services utilizing experienced consulting engineers and AFT tools. PMTG offers a solution to a wide variety of system analysis by building and investigating the models and providing direction on system changes and appropriate operation. Visit PMTG on the web: <https://www.pmtechgroup.com/>, or call at (719) 687-9145 for more information.

Related Topics

Technical Support.....	811
Contacting AFT.....	812
Information on Software Installation.....	813
Chempak Database Installation.....	813
Special File Control Features.....	816
Converting Models from Earlier Versions.....	817

Contacting AFT

- **Telephone** - (719) 686-1000
- **Fax** - (719) 686-1001
- **Online** - <https://www.aft.com/> download the latest maintenance releases, find out what is happening at AFT, and get all the latest information.
- **E-mail Support** - support@aft.com
- **Mail** - 2955 Professional Place (Suite 301), Colorado Springs, CO 80904, USA

Related Topics

Technical Support.....	811
Technical Support Options.....	811
Information on Software Installation.....	813
Chempak Database Installation.....	813
Special File Control Features.....	816
Converting Models from Earlier Versions.....	817

Information on Software Installation

For information on Local and Server installations of AFT software, please visit our website at <https://www.aft.com/support/installation>.

Related Topics

Technical Support.....	811
Technical Support Options.....	811
Contacting AFT.....	812
Chempak Database Installation.....	813
Special File Control Features.....	816
Converting Models from Earlier Versions.....	817

Chempak Database Installation

AFT Fathom does not include the [Chempak Fluid Library](#) by default. The Chempak Database is a separate product that must be downloaded and installed as an addition to AFT Fathom to access the Chempak Fluid Library in the software.

Chempak File Location

The Chempak Data is installed in **C:\AFT Products\Chempak Data** by default. If both AFT Fathom and the Chempak Database are installed to their default locations, no further configuration is necessary. However, if AFT Fathom or the Chempak Database are not installed in a default location, AFT Fathom will need to know where to look for the Chempak Database. This is specified in the [control.aft file](#), located in the same folder as Fathom.exe. If the files cannot be found the following message is displayed:

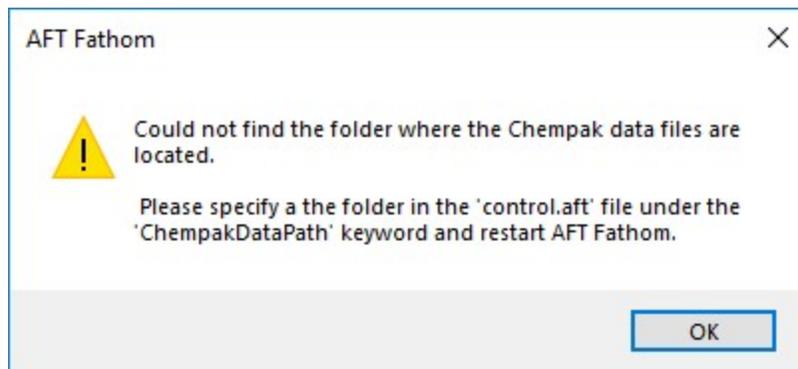
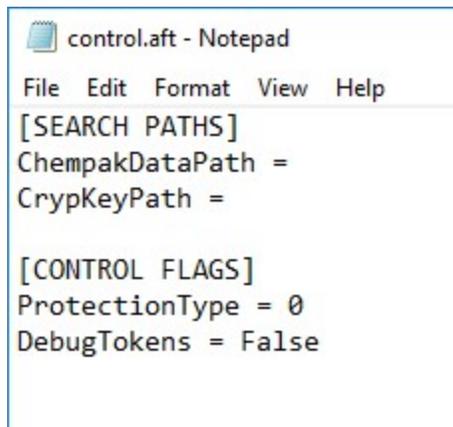


Figure 1: AFT Fathom cannot find the Chempak Database

Control.aft can be edited with a text editor, and by default looks like:



```
control.aft - Notepad
File Edit Format View Help
[SEARCH PATHS]
ChempakDataPath =
CrypKeyPath =

[CONTROL FLAGS]
ProtectionType = 0
DebugTokens = False
```

Figure 2: Default control.aft file

If the Chempak Database was installed to a location other than the default, the line **ChempakDataPath =** will need to be edited to include the directory it was installed to. For example: **ChempakDataPath = C:\My AFT Files\Chempak Data**

Upgrading Chempak Database Version

When upgrading to a new version of AFT Fathom, you may receive a message that the database is outdated.

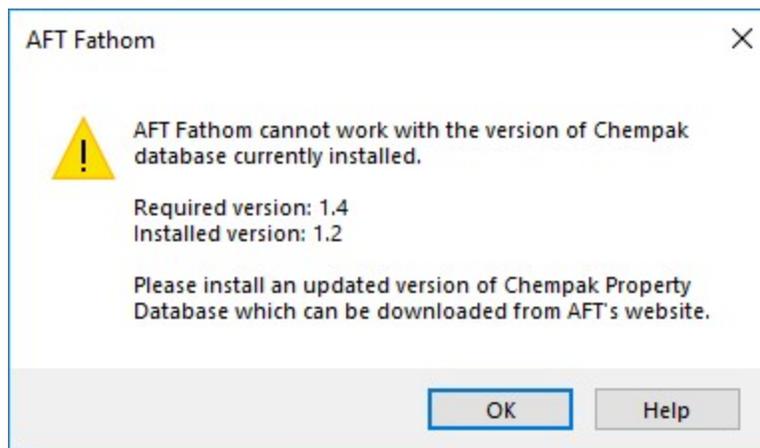


Figure 3: Outdated Chempak database

To fix this issue, download the newest version of the Chempak Database from the AFT website. Be sure to install to the same location as the existing Chempak Database, and double check the control.aft file if necessary.

Related Topics

Technical Support.....	811
Technical Support Options.....	811
Contacting AFT.....	812
Information on Software Installation.....	813
Special File Control Features.....	816
Converting Models from Earlier Versions.....	817

Output Files

Numerical results and output from the Solver are saved in Output Files (*.out file extension). Output Files are stored in the same location as the model file, and a unique file will exist for each [scenario](#) that is run, differentiated by numbers appended to the file name. The General section of the [Output window](#) will list which Output File is used for the current scenario.

The Output File will only remain on disk if the model file is saved after it has been run.

Changing various input data parameters for the model will invalidate the previous results, and a prompt will appear stating: "This action will clear all results and delete the output file for the current scenario and any children. Do you want to continue the action?"

Renaming the model file in the Windows File Explorer will unlink the model file from any existing Output Files. Instead, open the model file and choose File > Save As which will rename and copy all Output Files. Alternatively, the Save Scenario to File options can be used from the Scenario Manager.

In the [XTS Module](#), the file size will be heavily dictated by the simulation duration and time step specified on the [Simulation Duration panel](#) in Analysis Setup.

Updated Fluids Library

If the data in a shared fluid library is changed outside of the model you are opening, the following informational message will appear when opening the file:

The following fluid(s) used in the scenario have different property data than what is currently in the library:

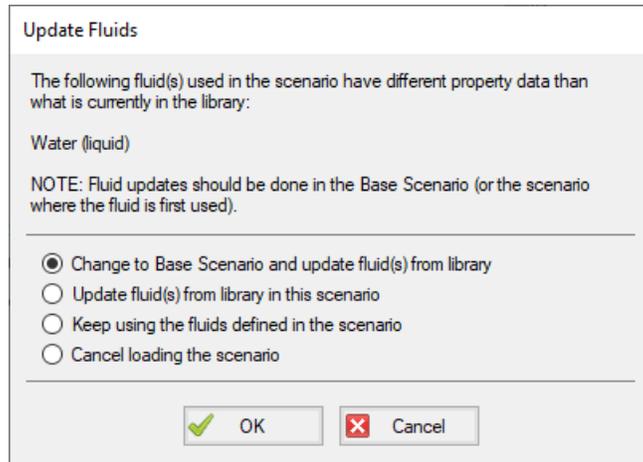


Figure 1: Update Fluids dialog window which appears if the fluid properties for the model have changed since it was last opened

There are a few reasons that the fluid data may have been changed:

1. The model was defined in a previous version of Fathom using a fluid from the [AFT Standard](#) fluid library. The AFT Standard fluid library was updated in AFT Fathom 12 in order to standardize the reference data. If the model uses the same fluid properties for all scenarios, it is recommended to use the "Change to Base Scenario and update..." option to update the fluid properties throughout the model. If different scenarios use different fluid properties then it is recommended to use the "Update fluid(s) from library in the scenario" option to update each scenario individually.
2. The model is defined with a custom fluid in the [Fluid Library](#) and the custom fluid was modified outside of this model. If the fluid was changed in error, then choose the "Keep using the fluids defined in the scenario" option or cancel loading the model and change the fluid. If the changes were intended then update the fluid data used in the model using the "Change to Base Scenario and update..." option.

Special File Control Features

What is the Control.aft File?

Control.aft is read as AFT Fathom begins to load and instructs AFT Fathom where certain key files and copy protection tokens are located. Typically there is little need to change the settings, however this file can be used to override the internal defaults and customize your working environment.

Control.aft is typically located in the application folder. However, on networks, it may be necessary to locate it in the Windows\System folder of each client. If AFT Fathom finds the file in the Windows\System folder it will use it and ignore any file in the application folder.

Control.aft can be edited using any text-based editor. Notepad, supplied as part of the Windows installation, is a good choice. Be careful to avoid changing any of the headings or key words. If these are changed AFT Fathom will not find the important data.

Specifying Where Chempak is Located

Before AFT Fathom can use the [Chempak Fluid Database](#) it needs to know where the data is located. The path is specified in the control.aft file.

When Chempak is installed the setup procedure automatically updates any control.aft file specified with the path to the data.

You can manually edit the file by opening it in Notepad (or any text editor) and adding the following line to the [SEARCH PATHS] section (replace C:\AFT Products\Chempak Data with the path information based on your system):

```
ChempakDataPath = C:\AFT Products\Chempak Data
```

This will instruct AFT Fathom to look in the C:\AFT Products\Chempak Data directory for the data.

Also see [Chempak Data Installation](#).

Related Topics

Technical Support.....	811
Technical Support Options.....	811
Contacting AFT.....	812
Information on Software Installation.....	813
Chempak Database Installation.....	813
Converting Models from Earlier Versions.....	817

Converting Models from Earlier Versions

Models from earlier versions of AFT Fathom can be used with this version. In most cases, the model file can be opened in the newer version and automatically converted to the updated format. It is considered good practice to do a complete check on the model to ensure the conversion process was complete and accurate. In some cases, especially with older models (two or more versions behind), the automatic conversion is not successful. If this occurs, please contact our [support team](#) with a copy of the original model file.

Other files such as libraries (previously called "databases") or preference files can also be converted.

The [Workspace Layers Import Assistant](#) can be used for models created in versions prior to the existence of Workspace Layers. This tool allows views from Visual Report (which is now deprecated) and customized Workspace appearance to be preserved.

Related Topics

Technical Support.....	811
------------------------	-----

Technical Support

Technical Support Options.....	811
Contacting AFT.....	812
Information on Software Installation.....	813
Chempak Database Installation.....	813
Special File Control Features.....	816

Examples

Walk-Through Examples

The AFT Fathom Walk-Through Examples are designed to get new users trained to use AFT Fathom correctly in the shortest amount of time. If you are using AFT Fathom for the first time, start with the [Beginner - Three Reservoir Example](#) because it offers an in-depth explanation of how to use AFT Fathom. Otherwise, choose an example from the table below that you wish to learn more about. For examples using the AFT Fathom Add-on Modules, see the [AFT Fathom Modules Overview](#).

Completed versions of the example model files are included with the default installation: C:\AFT Products\AFT Fathom 13\Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
Beginner - Three Reservoir	Beginner	Water	3	-	A new AFT Fathom user should start here. Get in-depth instructions of model building by solving a classic three reservoir flow problem. Learn how to define a fluid in Analysis Setup, place pipes and junctions on the Workspace, define pipes and junctions, use the Model Data window, run a model, use the Output window, generate graphs in the Graph Results window, and view results in the Visual Report window.
Sizing a Pump	Beginner	Water	2	1	Learn how to size a pump to get the desired flowrate and head through an open system. Explore the differences between the Sizing and Pump Curve features in the Pump junction. See how to interpret results, select a pump curve, and generate a Pump vs. System curve graph.
Control Valve	Beginner	Water	2	-	Learn about the Control Valve junction and how to use the Valve Summary

Walk-Through Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
					tab in the Output window. See how to use the results to adjust the pipe size until reaching a desired minimum pressure drop across the control valve.
Heat Transfer in a Pipe	Beginner	Water	1	-	Learn how to enable heat transfer calculations in Analysis Setup and how to use the Convective Heat Transfer model on the Heat Transfer tab of the Pipe Properties window. See how to use the Output Control window to display relevant heat transfer output parameters.
Heat Exchanger System	Beginner	Water	7	1	Learn how to define a Resistance Curve in a Heat Exchanger junction and how to size a pump in a closed system. See how to use the Pump Summary and Heat Exchanger Summary tabs in the Output window and how to generate a Pump vs. System Curve in the Graph Results window.
Spray Discharge System	Beginner	Water	11	-	Learn how to define the Spray Discharge junction using the Cd Spray Loss Model. See how to use the Output Control window to display the discharged flow rate parameter. Finally, learn how to enable the Isometric Grid on the Workspace, route pipes on the grid, and rotate junctions.
Pump Sizing and Selection with Flow Control Valves	Beginner	Water	9	1	Learn how to resolve the Reference Pressure

Walk-Through Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
					Needed window that may appear when modeling pumps and flow control valves in series. See how to use the results to select a pump curve for the system.
Freon Delivery System with Heat Transfer	Beginner	R11 and R123	7	1	Learn how to enable heat transfer, use a NIST REFPROP fluid, and use the Convective Heat Transfer model in a Heat Exchanger junction. See how to view and interpret temperature results in the Pipes section of the Output window.
Cooling System	Beginner	Water	20	2	Learn how to import pump curve data to evaluate the performance of different pump curves. See how to use the Pump Summary tab in the Output window to view the power consumed by the pumps.
Housing Project	Beginner	Water	25	-	Learn how to use Scenario Manager and Global Pipe Edit to rapidly change pipes sizes to analyze different system configurations. Also see how Assigned Flow junctions and Special Conditions work.
Beginner - Cost Calculation	Beginner	Water	10	1	Learn how to add, connect, and use both engineering and cost libraries. See how to define the Cost Settings panel in Analysis Setup, include objects in the cost calculations, and view the Cost Report in the Output window.

Walk-Through Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
Non-Newtonian Phosphates Pumping	Beginner	Phosphate	2	1	Learn how to use and compare the Homogeneous Scale-up, Power Law, and Bingham Plastic non-Newtonian viscosity models in a simple system.
NFPA Firewater System	Beginner	Water	71	-	Learn how to enable NFPA calculations in the Miscellaneous group in Analysis Setup. Explore how to use Groups to create an NFPA Report. See where to find the NFPA Report in the Output window.
Crude Oil Pipeline	Intermediate	Viscous Fluid	3	1	Learn how to specify a User Defined Fluid. See how results change when using the Viscosity Correction on the Optional tab of the Pump Properties window. Explore the sensitivity of junction elevation of pump speed as it relates to pressure in the system.
Hot Water System	Intermediate	Water	24	3	Learn how to add, connect, and use a library to define a pump curve. See how to use Global Junction Edit to rapidly apply changes. Explore how to use the Pump Configurations feature and Scenario Manager to analyze the performance effects of impeller size and pump speed.
Plant Cooling Cost Calculation	Intermediate	Water	34	4	Explore how to create cost libraries for pipe materials, pipe fittings and losses, and junctions. Learn how to use the Library Manager window to connect lib-

Walk-Through Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
					aries. See how to interpret the Cost Report in the Output window.
Non-Newtonian Mill Discharge Slurry	Intermediate	Mill Discharge Slurry	1	-	Learn how to use raw rheological data to define the Bingham Plastic, Power Law, and Homogeneous Scale-up viscosity models. See how the various viscosity models produce different results using the Multi-Scenario Output feature.
Flow Induced Vibration Calculations	Intermediate	Water	7	-	Learn how to analyze the cause and concerns of Flow Induced Vibration (FIV) using the Excel Export Manager and Vibration Analysis Spreadsheet.

Beginner - Three Reservoir

Summary

This example provides an overview of AFT Fathom's layout and structure. The basic features will be used to build a three-pipe, four-junction model to solve the classic three-reservoir problem.

Topics Covered

- Drawing the system with pipes and junctions
- Entering pipe and junction properties
- Completing Analysis Setup

Required Knowledge

No previous experience with AFT Fathom is required to complete this example. It begins with the most basic elements of laying out the pipes and junctions and solving the system hydraulics with the Newton-Raphson methodology.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Beginner - Three Reservoir.fth***

Step 1. Start AFT Fathom

➤ **To start AFT Fathom**, click Start on the Windows taskbar, choose AFT Fathom 13, and launch the program. (This refers to the standard menu items created by setup. You may have chosen to specify a different menu item during installation).

As AFT Fathom starts, the start-up screen, as shown in Figure 1, appears with several options before to start building a model. Some of the actions available are:

- Open a recent model, browse to a model file, or browse to an Example
- Activate an Add-on Module
- Select ASME Water or a recently used fluid to be the Working Fluid
- Review or modify Modeling Preferences
 - Select a Unit System
 - Filter units to include Common Only or Common Plus Selected Industries
 - Choose a Grid Style
 - Select a default Pipe Material
- Access other Resources, such as Quick Start Guides, Help Files, and Video Tutorials

Note: If you are working through the Metric Units version of the Examples, be sure to specify either Metric Only or Both with Metric Defaults as the unit set by modifying the Modeling Preferences.

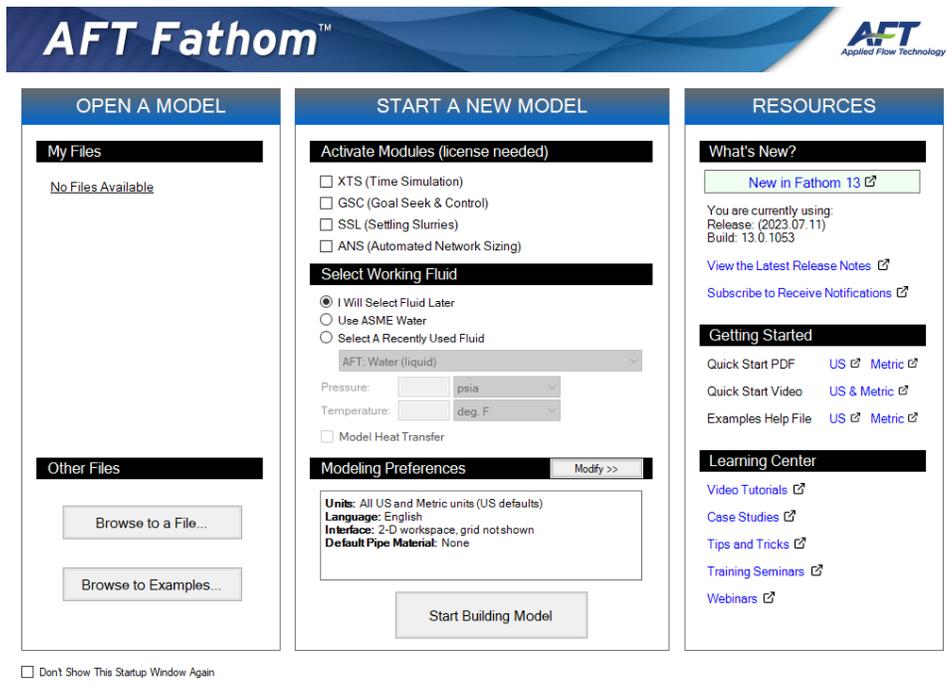


Figure 1: AFT Fathom 13 Start-Up Window

After clicking the Start Building Model button on the start-up window, the Workspace window is initially always the active (large) window, as seen in Figure 2. The tabs such as "Workspace" in the AFT Fathom window represent the Primary Windows. Each Primary Window contains its own toolbar that is displayed directly beneath the Primary Window tabs.

AFT Fathom supports dual monitor usage. You can click and drag any of the primary window tabs off of the main AFT Fathom window. Once you drag one of the primary windows off of the AFT Fathom window, you can move it anywhere you like on your screen, including onto a second monitor in a dual monitor configuration. To add the primary window back to the main Fathom primary tab window bar, simply click the X button in the upper right of the primary window.

To ensure that your results are the same as those presented in this documentation, this example should be ran using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

The Workspace window

The Workspace window is the primary vehicle for building your model. This window has three main areas: the Toolbox, the Quick Access Panel, and the Workspace itself. The Toolbox is the bundle of tools on the far left.

The [Quick Access Panel](#) is on the right. It is possible to minimize the Quick Access Panel by clicking on the thumbtack pin in the upper right of the Quick Access Panel in order to allow for greater Workspace area.

The [Workspace](#) takes up the rest of the window.

Walk-Through Examples

You will build your pipe flow model on the Workspace using the Toolbox tools. At the top of the Toolbox is the Pipe Drawing Tool and Annotation Tool. The Pipe Drawing tool, on the upper left, is used to draw new pipes on the Workspace. The Annotation tool allows you to create annotations and auxiliary graphics.

Below the two drawing tools are twenty-three icons that represent the different types of junctions available in AFT Fathom. Junctions are objects that connect pipes and also influence the pressure or flow behavior of the pipe system. The twenty-three junction icons can be dragged from the Toolbox and dropped onto the Workspace.

When you pass your mouse pointer over any of the Toolbox tools, a tooltip identifies the tool's function.

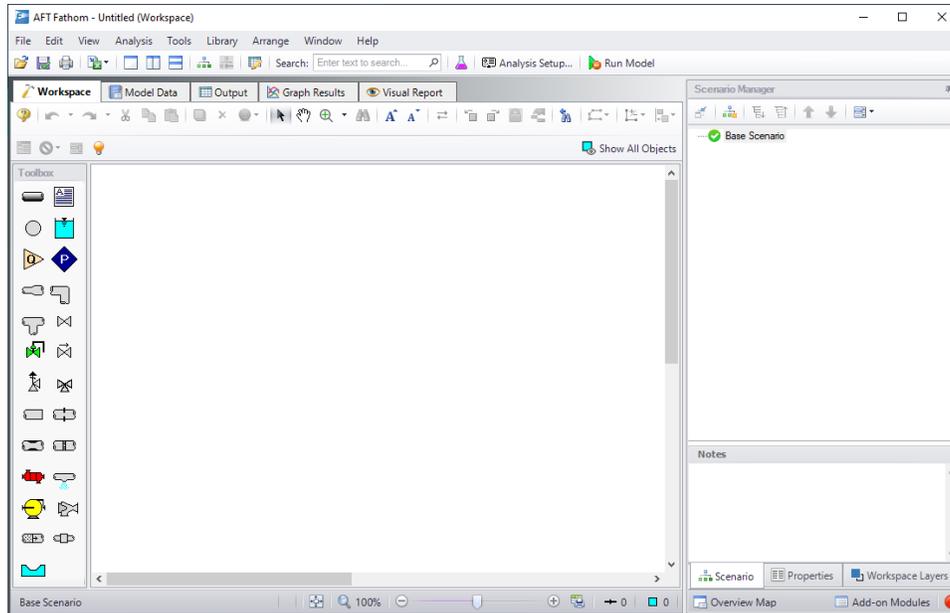


Figure 2: The Workspace window is where the model is built

Step 2. Complete the Analysis Setup

➤ **Next, click Analysis Setup on the Toolbar** that runs across the top of the AFT Fathom window. This opens Analysis Setup (see Figure 3). Analysis Setup contains seven groups (additional groups will be displayed if the GSC, XTS or ANS modules are active). Each group needs to be completed (indicated with a green checkmark next to the group name) before AFT Fathom allows you to run the Solver.

Analysis Setup can also be accessed by clicking on the Status Light on the [Quick Access Panel](#). Once the Analysis Setup is complete, the Model Status light turns from red to green.

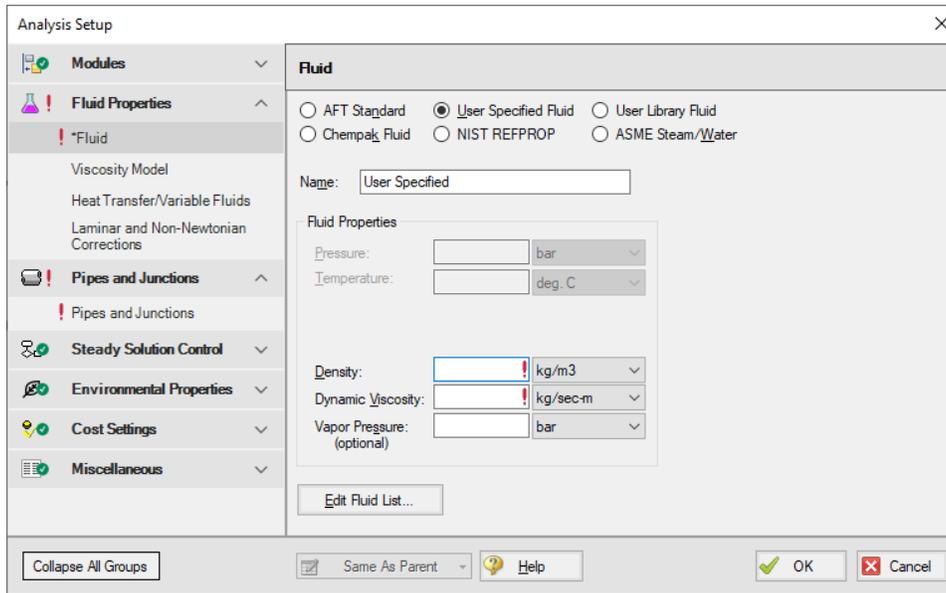


Figure 3: The Analysis Setup tracks the model's status and allows users to specify required parameters

A. Define the Modules Group

Although Analysis Setup initially opens to the Fluid panel of the Fluid Properties group, the first group, Modules, always has a green check when you start AFT Fathom because there are no modules activated by default. No further input is required here for this example.

B. Define the Fluid Properties Group

Next is the Fluid Properties group. This group allows you to specify your fluid properties (density, dynamic viscosity, and optional vapor pressure), viscosity model, heat transfer, and fluid corrections. Start by clicking on Fluid to open the **Fluid panel** (see Figure 4).

You can model the fluid properties using the following data sources:

1. **AFT Standard:** This fluid model uses fluid data from the AFT Standard library. These fluid properties are either temperature dependent or dependent on the solids concentration. Type in the desired condition (e.g., temperature), click the Calculate Properties button and the required properties are calculated. Users can add their own fluids to this option. This model supports heat transfer analysis if specific heat and thermal conductivity data is included. Custom fluids are created by opening the Library Manager window from the Library menu or by clicking the Edit Fluid List button on the Fluid panel.
2. **User Specified Fluid:** This fluid model allows you to directly type in the density, viscosity and vapor pressure. You cannot perform heat transfer analysis with a User Specified Fluid fluid.
3. **User Library Fluid:** This fluid model allows you to select user created fluids that are stored in the user's Fluid Library. You can perform heat transfer analysis with a User Library Fluid fluid if

specific heat, thermal conductivity, enthalpy data has been included in the fluid properties.

4. **ASME Steam/Water:** This fluid model obtains water data from the ASME Steam tables built into AFT Fathom. It also supports steam data, which can be used in AFT Fathom if incompressible.
5. **NIST REFPROP:** This fluid model allows you to select a single fluid or create a mixture of fluids from the REFPROP list. These fluid properties are pressure and temperature dependent, although some are temperature dependent only. This fluid model supports heat transfer analysis.
6. **Chempak Fluid:** This fluid model allows you to select a single fluid or create a mixture of fluids from the Chempak list. These fluid properties are pressure and temperature dependent, although some are temperature dependent only. This fluid model supports heat transfer analysis. Chempak is an optional add-on.

After selecting ASME Steam/Water, NIST REFPROP, or Chempak Fluid, you must then select a Fluid Phase: Liquid or Gas. Then, upon entering a pressure, AFT Fathom will display a temperature range applicable to liquid or gas, depending upon which was selected.

Note: Because heat transfer has a significant impact on compressible flow fluid properties and AFT Fathom calculates flow on an incompressible basis, heat transfer is disabled when Gas is selected as the Fluid Phase.

➤ **Select AFT Standard**, by clicking the radio button next to the name. Then choose Water (liquid) from the list and click Add to Model. The properties for AFT Standard water are given only as a function of temperature. Enter 21 deg. C in the temperature box and click the Calculate Properties button.

➤ **Check the groups on the left side of Analysis Setup** and you should now see the second group is checked complete with green checkmarks.

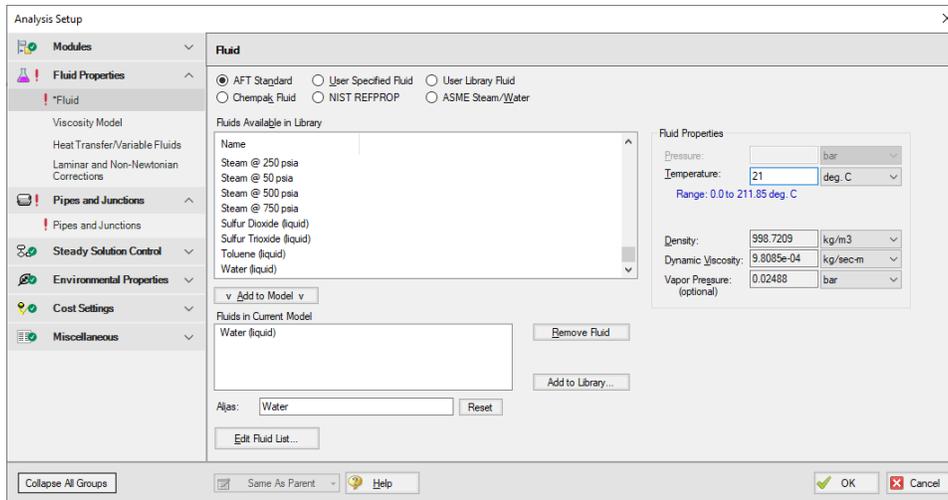


Figure 4: The Fluid panel lets you enter physical properties

On the **Viscosity Model panel**, users can specify how AFT Fathom will treat viscosity. By default, Newtonian is the viscosity model selected. AFT Fathom offers a variety of non-Newtonian models. However, for this example, the default selection of Newtonian will suffice.

On the **Heat Transfer/Variable Fluids panel**, users can specify variable fluid properties or enable heat transfer. For models with variable fluid properties, the values for density and viscosity are default fluid

properties. You can then enter different property values, if desired, for any pipe in the Pipe Properties window. Fluids that allow heat transfer can be modeled by choosing the Heat Transfer with Energy Balance option. In this case, you will be required to also enter heat transfer information for the pipes and temperature for some of the junctions. By default, Constant Fluid Properties is selected and will suffice for this example.

On the **Laminar and Non-Newtonian Corrections panel**, users can specify the type of corrections that the solver will apply when flow is laminar or if a non-Newtonian viscosity model is selected. The default selections are applicable for most cases, so no input is required for this example.

C. Define the Pipes and Junctions Group

In order to fully define this group, there needs to be pipes and junctions on the Workspace. To lay out the classic three-reservoir model, you will place the three reservoir junctions and a branch junction on the Workspace. Then you will connect the junctions with pipes.

To go back to the Workspace and save the inputs made in Analysis Setup, click OK.

I. Place the first reservoir

➤ **To start, click on and drag a reservoir junction from the Toolbox** and drop it on the Workspace. Figure 5 shows the Workspace with one reservoir.

Objects and ID numbers

Items placed on the Workspace are called *objects*. All objects are derived directly or indirectly from the Toolbox. AFT Fathom uses three types of objects: *pipes*, *junctions*, and *annotations*.

All pipe and junction objects on the Workspace have an associated ID number. For junctions, this number is, by default, placed directly above the junction and prefixed with the letter J. Pipe ID numbers are prefixed with the letter P. You can optionally choose to display either or both the ID number and the name of a pipe or junction. You also can drag the ID number/name text to a different location to improve visibility.

The reservoir you placed on the Workspace will take on the default ID number of 1. You can change this to any desired number greater than zero and up to 99,999.

Editing on the Workspace

Once on the Workspace, junction objects can be moved to new locations and edited with the features on the Edit menu. Cutting, copying, and pasting are all supported.

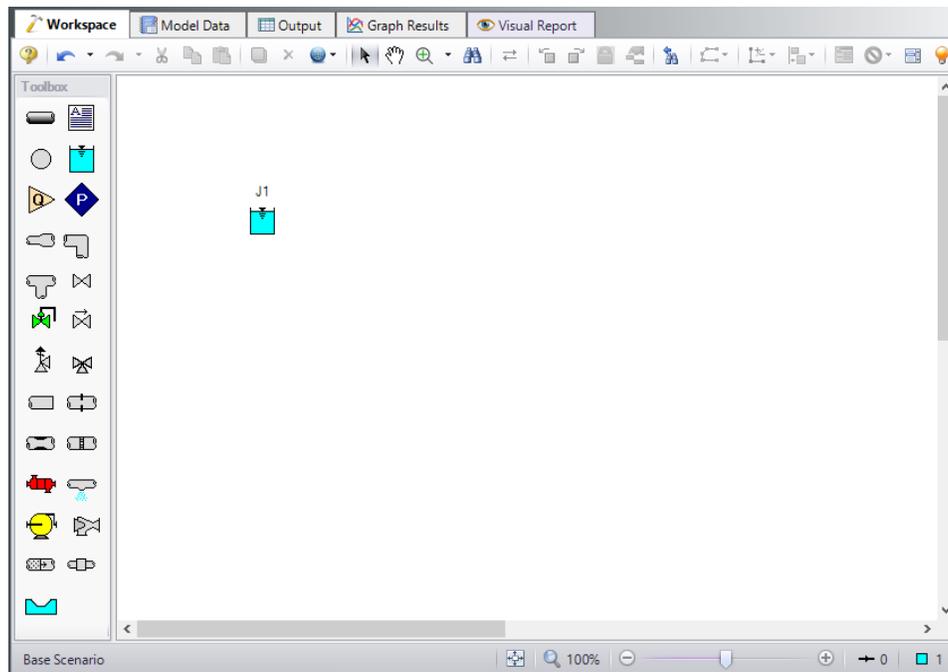


Figure 5: Three Reservoir Problem with one reservoir placed

II. Place the second and third reservoirs

The remaining two reservoirs can be created the same way as the first one or they can be derived from the existing reservoir.

➤ **To create a second reservoir from the existing one**, select junction J1 by clicking it with the mouse. A red outline will surround the junction. Choose Duplicate from the Edit menu and move the J2 junction to the right of J1. Your Workspace should appear similar to that shown in Figure 6.

If you like, you can Undo the Duplicate operation and then Redo it to see how these editing features work. To undo an operation, click on the undo button on the Toolbar or choose Undo from the Edit menu. To redo an operation, choose Redo from the Edit menu.

➤ **To create the third junction**, select one of the two reservoirs on the Workspace and once again choose Duplicate from the Edit menu. Arrange the three junctions, numbered J1, J2, and J3, as shown in Figure 7.

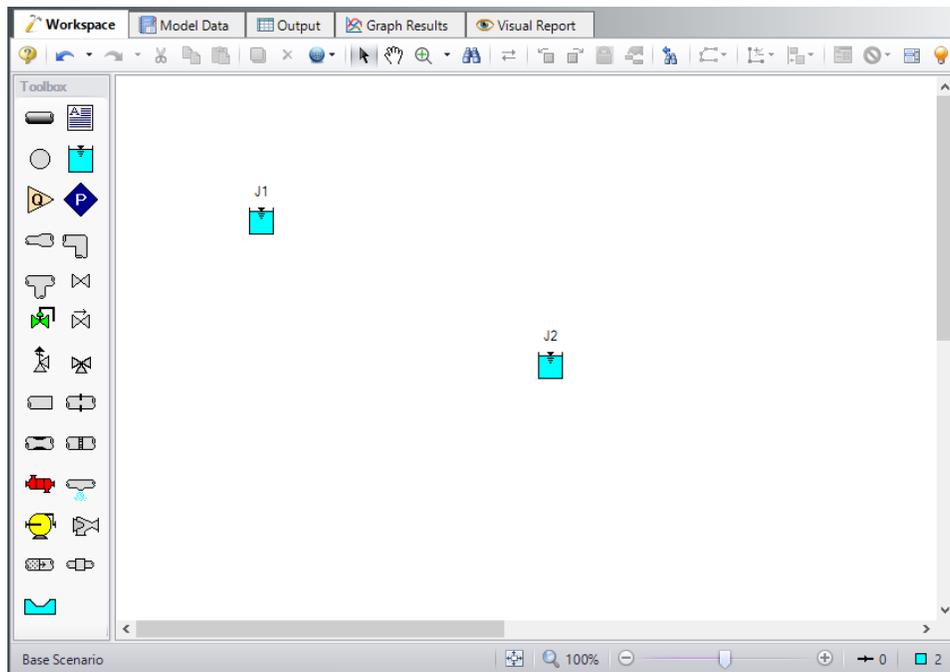


Figure 6: Three Reservoir Problem with two reservoirs placed

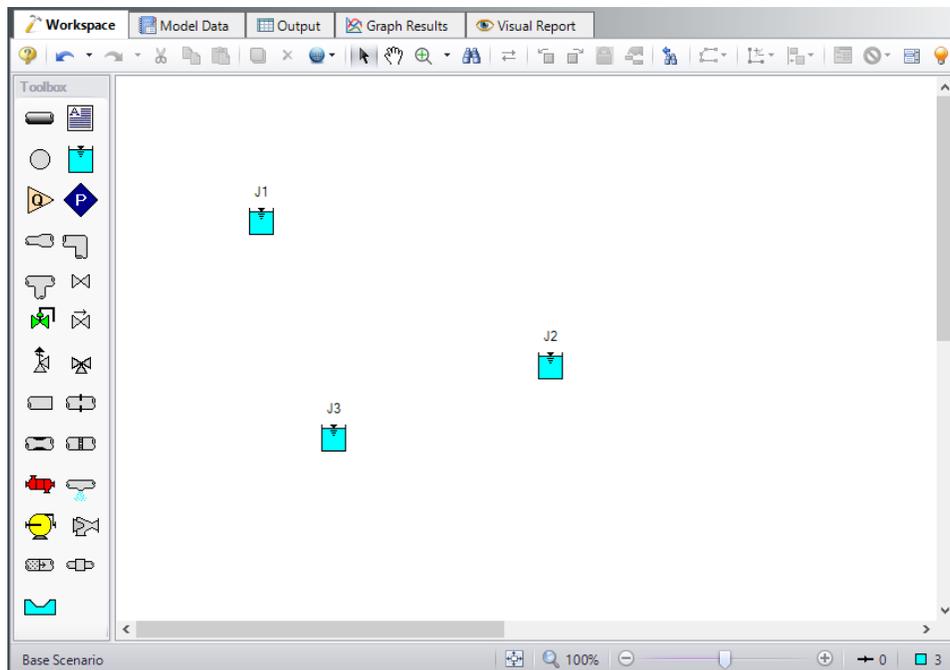


Figure 7: Three Reservoir Problem with three reservoirs placed

III. Place a branch junction

➤ **To add a Branch junction**, select a Branch from the Toolbox and place it on the Workspace as shown in Figure 8. The Branch will be assigned the default number J4.

Note: The relative location of objects in AFT Fathom is not important. Distances and heights are defined through dialog boxes. The relative locations on the Workspace establish the *connectivity* of the objects, but have **no bearing on the actual length or elevation relationships**.

➤ **Before continuing, save the work you have done so far.** Choose Save As from the File menu and enter a file name (Three Reservoir Problem, perhaps) and AFT Fathom will append the *.fth* extension to the file name.

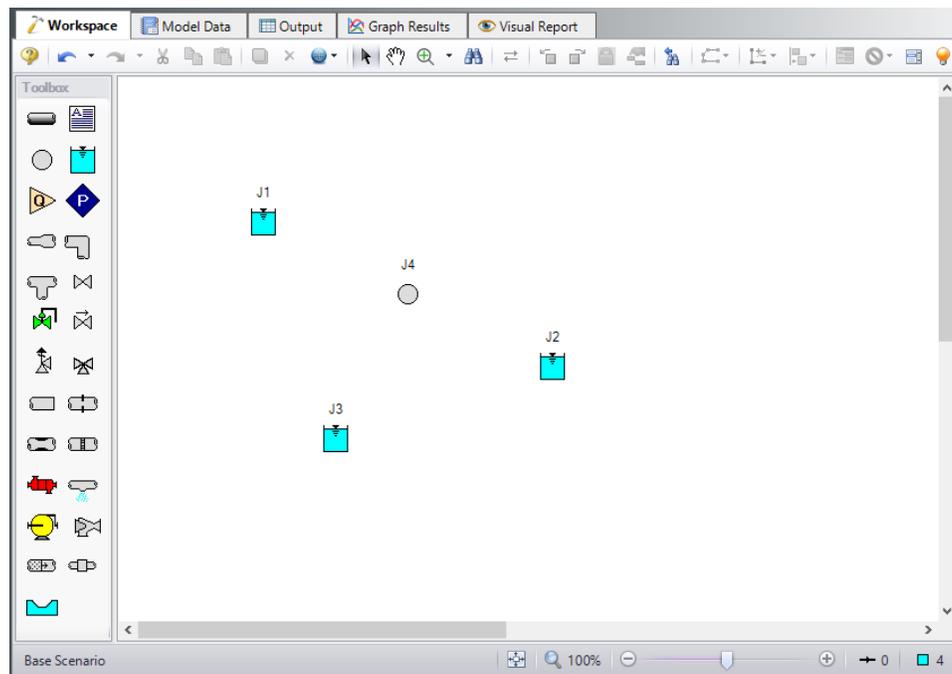


Figure 8: Three Reservoir Problem with all junctions placed

IV. Draw a pipe between J1 and J4

Now that you have four junctions, you need to connect them with pipes.

➤ **To create a pipe**, click the Pipe Drawing tool icon. The pointer will change to a crosshair when you move it over the Workspace. Draw a pipe above the junctions, similar to that shown in Figure 9.

The pipe object on the Workspace has an ID number (P1) shown near the center of the pipe.

➤ **To place the pipe between J1 and J4**, use the mouse to grab the pipe in the center, drag it so that its left endpoint falls within the J1 Reservoir icon, then drop it there (see Figure 10). Next, grab the right endpoint of the pipe and stretch the pipe, dragging it until the endpoint terminates within the J4 Branch icon (see Figure 11).

Walk-Through Examples

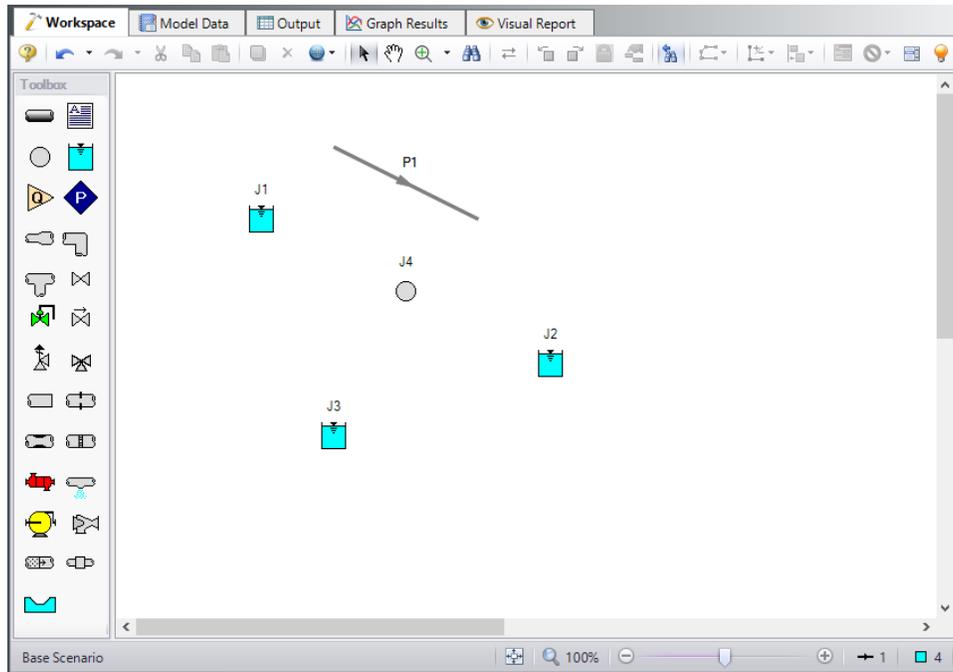


Figure 9: Three Reservoir Problem with first pipe drawn

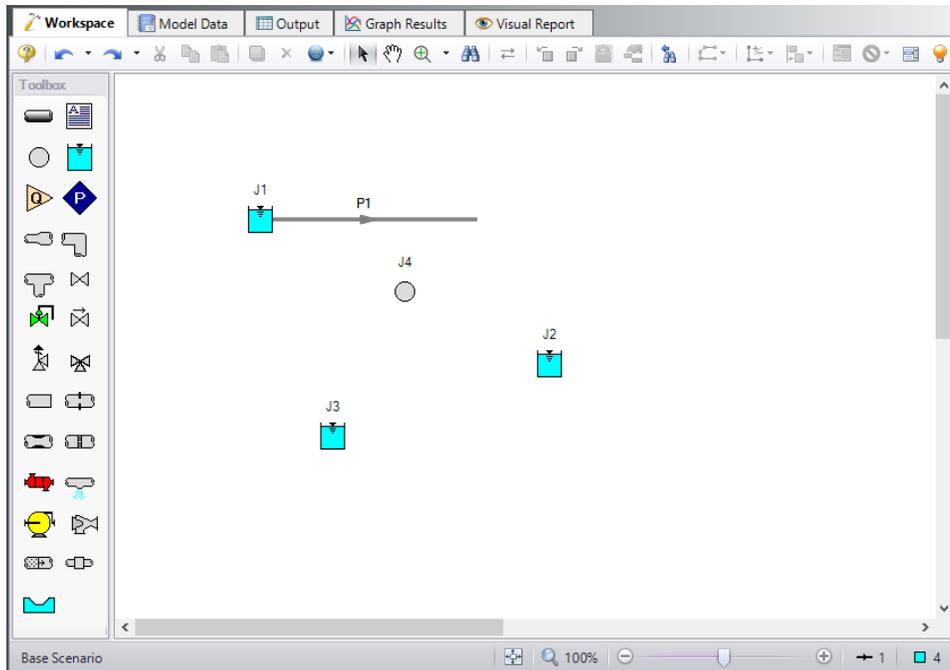


Figure 10: Three Reservoir Problem with first pipe inlet end connected

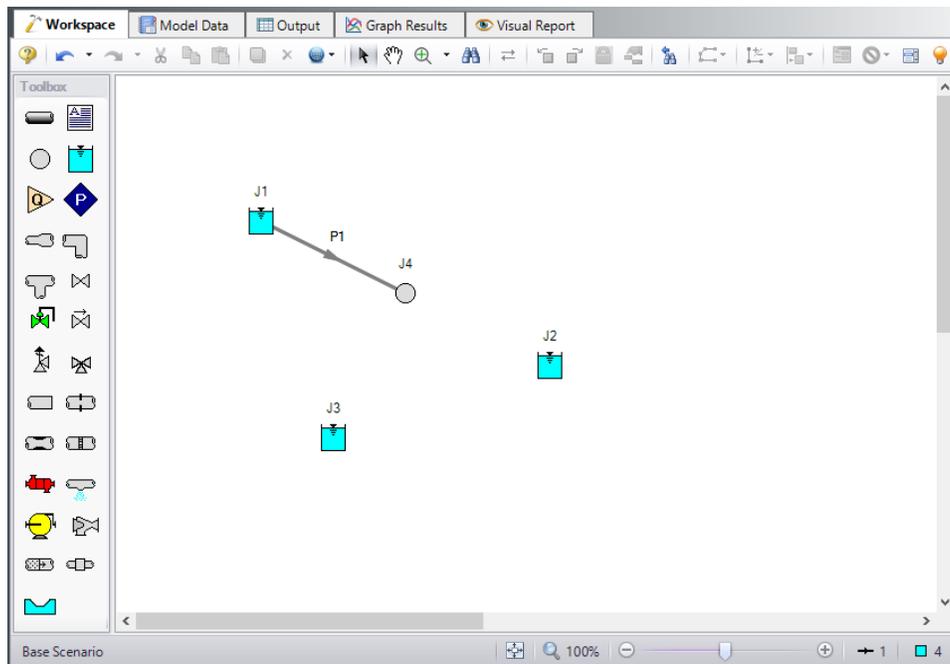


Figure 11: Three Reservoir Problem with first pipe connected

Reference positive flow direction

Located on the pipe is an arrow that indicates the reference positive flow direction for the pipe. AFT Fathom assigns a flow direction corresponding to the direction in which the pipe is drawn. You can reverse the reference positive flow direction by choosing Reverse Direction from the Arrange menu or selecting the reverse direction button on the Workspace Toolbar.

In general, the reference positive flow direction indicates which direction is considered positive. However, when used with pumps and certain other junction types the pipes must be in the correct flow direction because that is how AFT Fathom determines which side is suction and which is discharge. If the reference positive direction is the opposite of that obtained by the Solver, the output will show the flow rate as a **negative number**.

V. Add the remaining pipes

A faster way to add a pipe is to draw it directly between the desired junctions.

➤ **Activate the pipe drawing tool again** (double-click to allow pipe drawing tool to stay active). Position the cursor on the J4 Branch. Press and hold the left mouse button. Stretch the pipe to the J2 Reservoir then release the mouse button. Then draw a third pipe from the J4 Branch to the J3 Reservoir. Your model should now look similar to Figure 12. In Figure 12, the J3 label has been moved slightly to the left (using drag-and-drop) so it is more visible.

At this point all the objects in the model are graphically connected. Save the model by selecting Save from the File menu or Toolbar.

Note: It is generally desirable to lock your objects to the Workspace once they have been placed. This prevents accidental movement and disruption of the connections. You can lock all the objects by choosing Select All from the Edit menu, then selecting Lock Object from the Arrange menu. The lock button on the Toolbar will appear depressed indicating it is in an enabled state, and will remain so as long as any selected object is locked. Alternatively, you can use the grid feature enabled from the Arrange menu or the User Options under the Tools menu bar and specify that the pipes and junctions snap to grid.

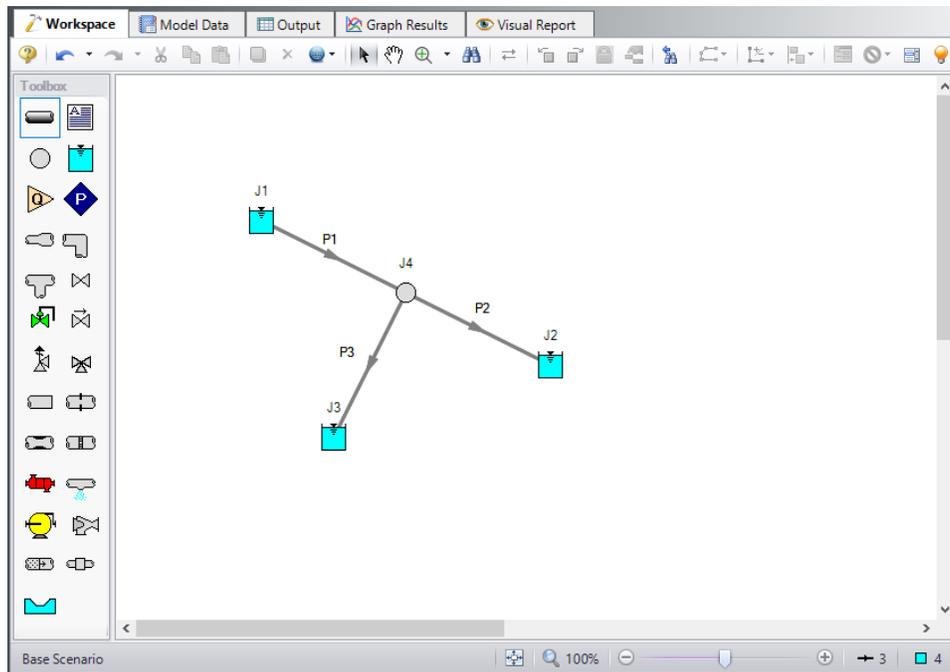


Figure 12: Three Reservoir Problem with all objects connected

VI. Define the pipes and junctions

To fully define the Pipes and Junctions group in Analysis Setup, all pipes and junctions must be connected and have the proper input data.

Object status

Every pipe and junction has an object status. The object status tells you whether the object is defined according to AFT Fathom's requirements. To see the status of the objects in your model, click the floodlight icon on the Workspace Toolbar (alternatively, you could choose Show Object Status from the View menu). Each time you click the floodlight, Show Object Status is toggled on or off.

When Show Object Status is on, the ID numbers for all undefined pipes and junctions are displayed in red on the Workspace. Objects that are completely defined have their ID numbers displayed in black. (These colors are configurable through User Options from the Tools menu.)

Because you have not yet defined the pipes and junctions in this sample problem, all the objects' ID numbers will change to red when you turn on Show Object Status.

Undefined Objects window

The Undefined Objects window lists all undefined pipes and junctions and further displays the items that are not yet defined.

VII. Define Reservoir J1

➤ **To define the first reservoir**, open the J1 Reservoir Properties window (see Figure 13) by double-clicking the J1 icon. Enter a reservoir Liquid Surface Elevation of 90 meters. You can assign any unit of length found in the adjacent dropdown list of units.

Note: You can also open an object's Properties window by selecting the object (clicking on it) and then either pressing the Enter key or clicking the Open Pipe/Jct Window icon on the Toolbar.

➤ **Enter a Liquid Surface Pressure of 0 barG (0 kPa (g))** and a Pipe Depth of 0 meters.

Note: You can specify preferred units for many parameters (such as meters for length) in the User Options window.

You can give the object a name, if desired, by entering it in the Name field at the top of the window. In Figure 13, the name of this reservoir is Supply Tank A. By default the junction's name is the junction type. The name can be displayed on the Workspace, Model Data or in the Output.

Most junction types can be entered into a custom library allowing the junction to be used multiple times or shared between users. To select a junction from the custom library, choose the desired junction from the Library list. The current junction will get the properties from the library junction.

The Copy Data From Jct list will show all the junctions of the same type in the model. This will copy the selected parameters from an existing junction in the model to the current junction.

The pipe table on the **Pipe Depth & Loss Coefficients** tab allows you to specify entrance and exit loss factors for each pipe connected to the reservoir (in this case there is one). You can enter standard losses by selecting the option buttons at the right. The default selection is the Custom option with loss factors specified as zero. To later change the loss factors, click within the pipe table and enter the loss. You can also specify a depth for the pipe.

The **Optional** tab allows you to enter different types of optional data. You can select whether the junction number, name, or both are displayed on the Workspace. Some junction types also allow you to specify an initial pressure as well as other junction specific-data. The junction icon graphic can be changed, as can the size of the icon.

Design Alerts can be entered for most junctions, which are applied to the pressure loss calculations for the junction in order to give additional safety margin to the model.

Each junction has a tab for **Notes**, allowing you to enter text describing the junction or documenting any assumptions.

The **Status** tab will list undefined parameters of the junction.

Walk-Through Examples

The highlight feature displays all the required information in the Properties window in light blue as shown in Figure 13. The highlight is on by default. You can toggle the highlight off and on by double-clicking anywhere in the window or by pressing the F2 key. The highlight feature can also be turned on or off by selecting it on the User Options menu. For the remainder of the examples, we will keep the highlight feature turned off.

➤ **Click OK.** If Show Object Status is turned on, you should see the J1 ID number turn black again, telling you that J1 is now completely defined.

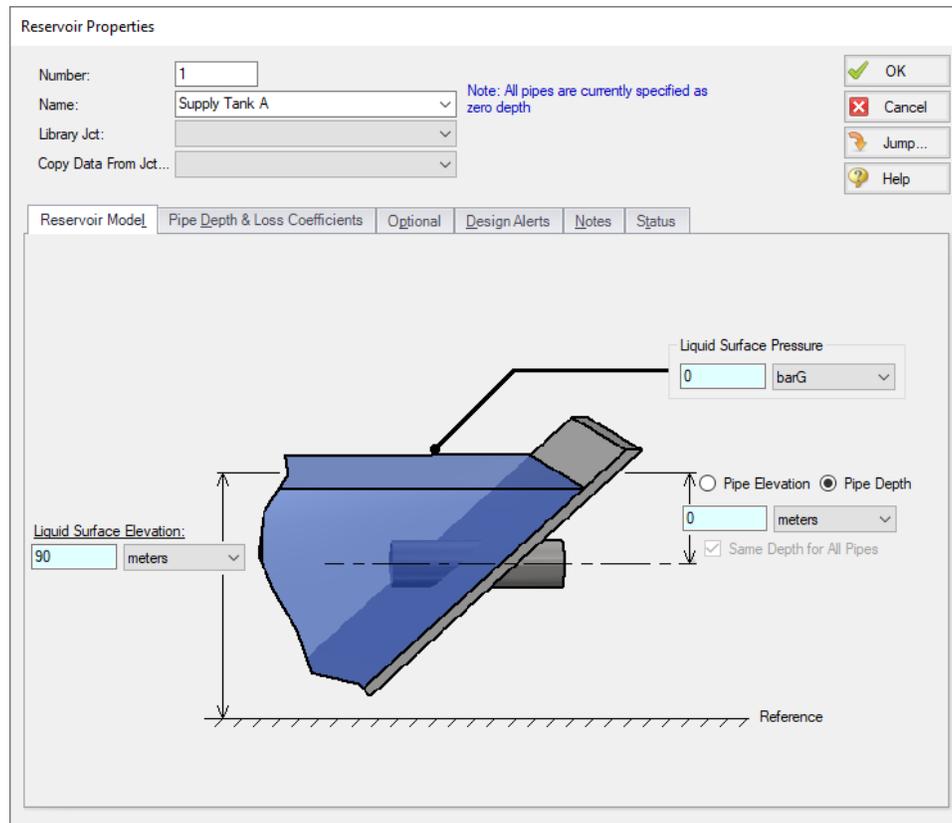


Figure 13: Properties window for Reservoir J1

The Inspection feature

You can check the input parameters for J1 quickly, in read-only fashion, by using the Inspection feature. Position the mouse pointer on J1 and hold down the right mouse button. An information box appears, as shown in Figure 14.

Inspecting is a faster way of examining the input in an object than opening the Properties window.

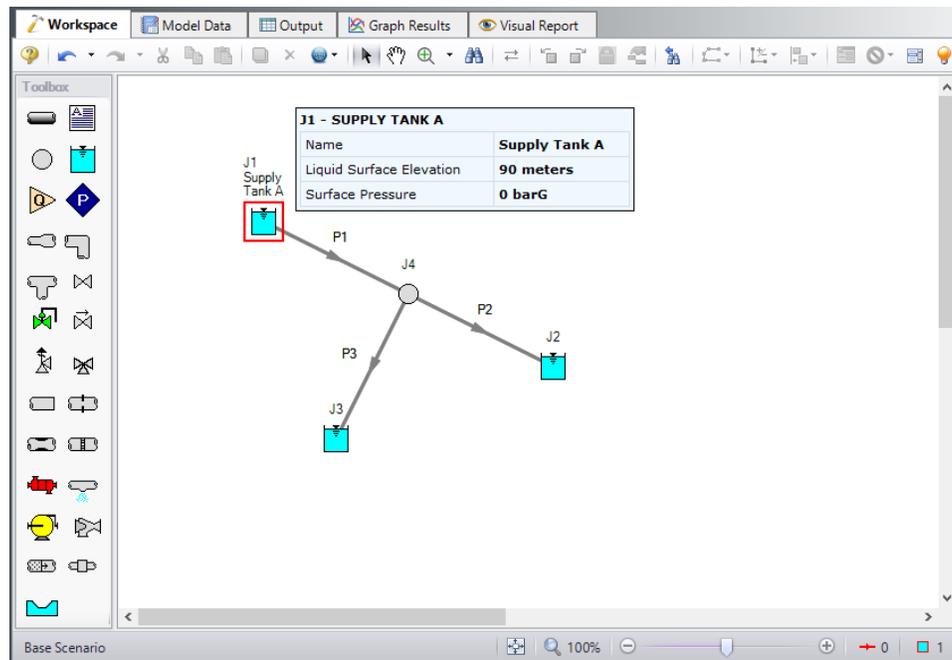


Figure 14: Inspecting from the Workspace with right mouse button

VIII. Define junctions J2, J3, and J4

1. J2 Reservoir
 - a. Liquid Surface Elevation = 60 meters
 - b. Liquid Surface Pressure = 0 barG
 - c. Pipe Depth = 0 meters (default)
2. J3 Reservoir
 - a. Liquid Surface Elevation = 30 meters
 - b. Liquid Surface Pressure = 0 barG
 - c. Pipe Depth = 0 meters (default)

➤ **Open the J4 Branch Properties window** (see Figure 15). In this window, all three connecting pipes should be displayed in the pipe table area. You could associate loss factors with each pipe by clicking within the pipe table and entering the data.

➤ **Enter an elevation of zero meters** for the J4 Branch junction (an elevation must be defined for all junctions). You could also click the Optional tab then specify a flow source or sink at the junction; for now leave this as zero. Click OK. All the junctions have now been defined.

➤ **Save the model** again before proceeding.

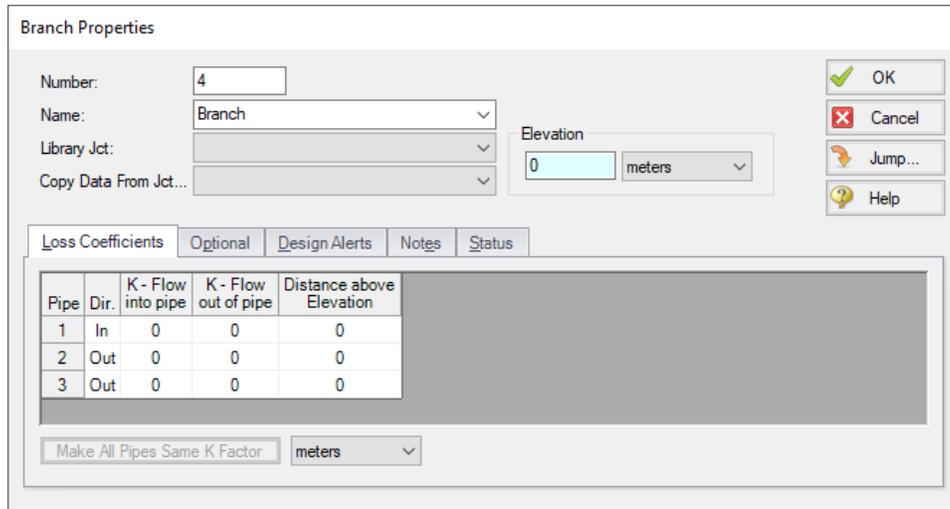


Figure 15: Properties window for Branch J4

IX. Define Pipe P1

Data for pipes and junctions can be entered in any order. In this example, we did the junctions first. The next step is to define all the pipes. To open the Pipe Properties window, double-click the pipe object on the Workspace.

➤ **First open the Pipe Properties window for Pipe P1** (see Figure 16). For Pipe P1, enter a length of 300 meters, select the Pipe Material as Steel - ANSI, Size of 8 inch , and Type as STD (schedule 40). Click OK. Pipe P1 is now defined.

The Pipe Properties window

The Pipe Properties window offers control over all important flow system parameters that are related to pipes.

The Inspect feature also works within the Pipe Properties window. To Inspect a connected junction, position the mouse pointer on the connected junction's ID number and hold down the right mouse button. This is helpful when you want to quickly check the properties of connecting objects. (You can also use this feature in junction Properties windows for checking connected pipe properties.)

By double-clicking the connected junction number, you can jump directly to the junction's Properties window. Or you can click the Jump button to jump to any other part of your model.

Figure 16: Properties window for Pipe P1

X. Define Pipes P2 and P3

Both pipes have a Pipe Material of Steel - ANSI, Size of 8 inch, and Type of STD (Schedule 40).

1. P2 Length = 600 meters
2. P3 Length = 900 meters

The Pipes and Junctions group should now be complete. Everything is ready to submit to the Solver.

➤ **Before running the model, save it one more time.** It is also a good idea to review the input using the Model Data window.

D. Define the remaining groups in Analysis Setup

Open Analysis Setup once more to confirm that all groups are defined. No inputs or alterations are required to complete the last three groups: Steady Solution Control, Environmental Properties, and Miscellaneous. All seven groups should have a green checkmark next to its name. Click OK.

D. Review input in the Model Data window

The Model Data window is shown in Figure 17. To change to this window, you can select it from the Primary Window tabs, Window menu, or pressing CTRL+M. The Model Data window gives you a text-

Walk-Through Examples

based perspective of your model. Selections can be copied to the clipboard and transferred into other Windows programs, saved to a formatted file, printed to an Adobe PDF, or printed out for review.

Data is displayed in three general areas. The top is called the general area, the middle the pipe area and the bottom the junction area.

The Model Data window allows access to all Properties windows by double-clicking the appropriate ID number in the far left column of the table. You may want to try this right now.

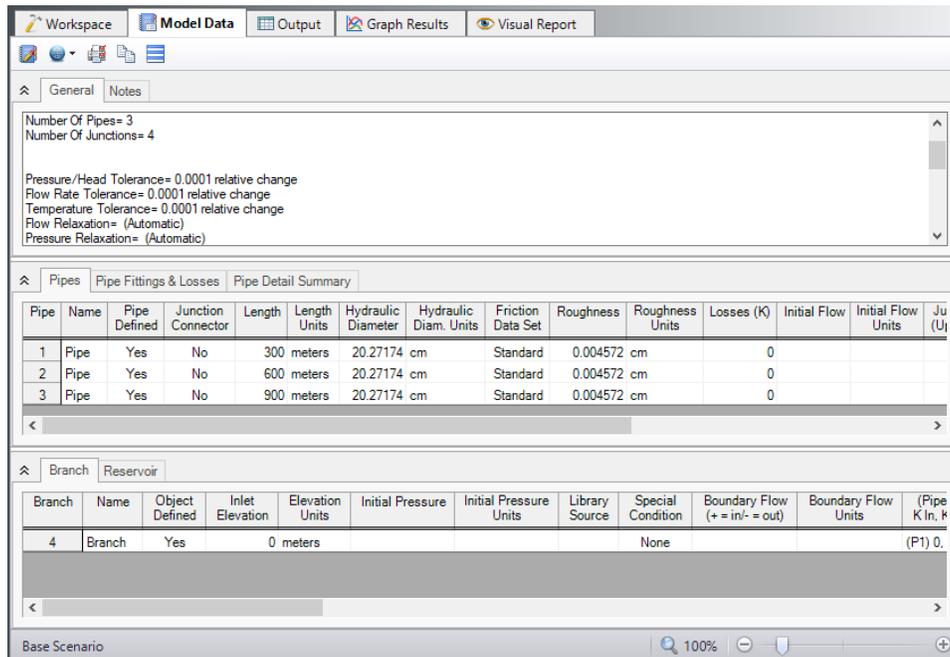


Figure 17: The Model Data window shows all input in text form

Step 3. Run the Solver

➤ **Click Run Model from the Toolbar** or from the Analysis Menu. During execution, the Solution Progress window is displayed (see Figure 18). You can use this window to pause or cancel the Solver's activity. When the solution is complete, the Output, Visual Report, and Graphs Results windows can be accessed directly from the Solution Progress window. Click the Output button and the text-based Output window will appear (see Figure 19). The information in the Output window can be reviewed visually, saved to file, exported to a spreadsheet-ready format, copied to the clipboard, printed to an Adobe PDF file, and printed out on the printer.

Walk-Through Examples

Solution Progress - Complete

Maximum Iterations: 50000 Run Time: .25

Relaxation: Automatic

	Absolute Tolerance Max Out of Tol.	Relative Tolerance Max Out of Tol.	Total Iterations
Head: 1.0E-04 Relative Change Not used (Absolute Change meters)	1.634E-04	6.705E-08	54
Vol. Flow Rate: 1.0E-04 Relative Change Not used (Absolute Change m3/hr)	2.093E-02	3.989E-05	4
Temperature:			

Setting up input...
 Determining connectivity...
 Initializing model...
 Running Solver...
 Writing output to file...
 Calculations complete.

Other Actions Pause Cancel Graph Results... Visual Report... Output...

Figure 18: The Solution Progress window displays the convergence progress

Workspace | Model Data | **Output** | Graph Results | Visual Report

General | Warnings | Design Alerts | Reservoir Summary

Execution Time= 0.30 seconds
 Total Number Of Head/Pressure Iterations= 54
 Total Number Of Flow Iterations= 4
 Total Number Of Temperature Iterations= 0
 Number Of Pipes= 3
 Number Of Junctions= 4
 Matrix Method= Gaussian Elimination
 Pressure/Head Tolerance= 0.0001 relative change

Pipes

Pipe	Name	Vol. Flow Rate (m3/hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)
1	Pipe	542.3	4.668	7.308	0.9045	90.00	0.00	-6.404	-6.404	-8.815	24.616	0.9045	7.3083	1.013	7.417
2	Pipe	171.3	1.474	7.406	1.0024	0.00	60.00	6.404	6.404	5.876	5.384	7.4062	1.0024	7.417	1.013
3	Pipe	371.0	3.193	7.366	0.9623	0.00	30.00	6.404	6.404	2.938	35.384	7.3661	0.9623	7.417	1.013

All Junctions | Branch | Reservoir

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Supply Tank A	1.013	1.013	1.013	1.013	542.3	150.45	0
2	Reservoir	1.013	1.013	1.013	1.013	171.3	47.52	0
3	Reservoir	1.013	1.013	1.013	1.013	371.0	102.93	0
4	Branch	7.369	7.369	7.417	7.417	N/A	N/A	0

Base Scenario 100%

Figure 19: The Output window displays output in text form

Step 4. Review the output

The Output window is similar in structure to the Model Data window. Three areas are shown, and you can minimize or enlarge each area by clicking the arrow next to the General, Pipes, and Junctions tabs or from the View menu.

A. Specify output control

By default, AFT Fathom has a predefined set of Output Control parameters that are specified and the default units used depend on which unit set was selected when first running AFT Fathom (see Figure 1). A default title is also assigned in the Output Control.

➤ **Open Output Control** from the Tools menu (Figure 20). Click the General tab, enter a new title (if you like you can title this Classic Three-Reservoir Problem), then click OK to accept the title and other default data.

In addition, this window allows you to select the specific output parameters you want in your output. You also can choose the units for the output.

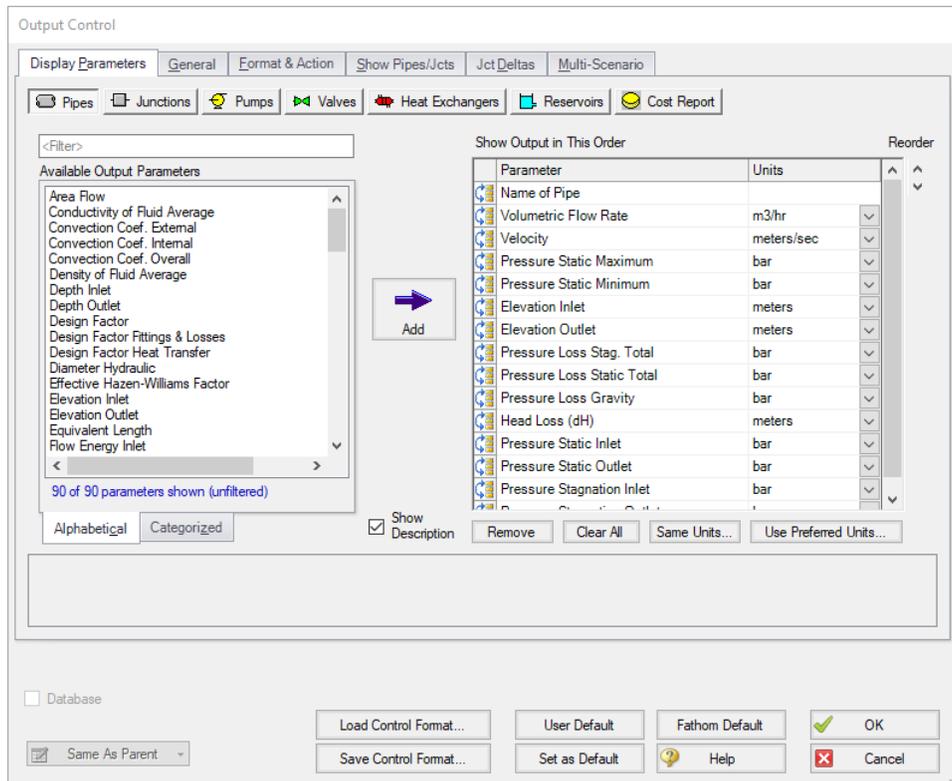


Figure 20: The Output Control window lets you customize the output

B. Modify the output format

If you selected the default AFT Fathom Output Control, the Pipe Results table will show volumetric flow rate in the second column (Figure 19) with units of m³/hr (cubic meters per hour).

- ➤ **Select Output Control from the Tools menu or Toolbar.** On the right side of the Pipe tab is the list of selected output parameters. Click Volumetric Flow Rate and change the units by selecting m³/min (cubic meters per minute) from the list at the bottom.
- ➤ **Click OK to display changes to the current results.** You should see the volumetric flow rate results, still in the second column, in units of m³/min. Notice the Velocity results in the third column.
- ➤ **Select Output Control from the Tools menu one more time.** The reorder icons on the left side of the Output List allows you to reorder parameters in the list.
- ➤ **Select the Velocity parameter** and use the Reorder scroll bar to move it up to the top of the parameter list.
- ➤ **Click OK to display the changes to the current results.** You will see in the Pipe Results table that the first column now contains Velocity and the third column contains the volumetric flow rate. The Output Control window allows you to obtain the parameters, units and order you prefer in your output. This flexibility will help you work with AFT Fathom in the way that is most meaningful to you, reducing the possibility of errors.
- ➤ **Lastly, double-click the column header Velocity in the Output window Pipe Results Table.** This will open a window in which you can change the units again if you prefer. These changes are extended to the Output Control parameter data you have previously set.

C. View the Output on the Workspace with Workspace Layers

- ➤ **Return to the Workspace window** by choosing it from the Window menu, clicking on the Workspace tab on the tool bar, or by pressing CTRL+W. Workspace Layers allows you to integrate your text results with the graphic layout of your pipe network.
- ➤ **Select the Workspace Layers tab** at the bottom right of the screen within the Quick Access Panel. Select the layer named All Objects Layer and then select the single gear icon above to edit the Layer Settings. Alternatively, while hovering over the layer name, the Layer Settings icon will be revealed to the right. Select the Pipe Parameters tab and search for Volumetric Flow Rate with the Filter or within the Pipe Parameters list. Double click Volumetric Flow Rate to add it to the Workspace Layer. Select the Junction Parameters tab and search for both Junction Name and the Pressure Static Inlet. After adding these parameters to the display list, press Close. Rearrange the labels as desired for readability.

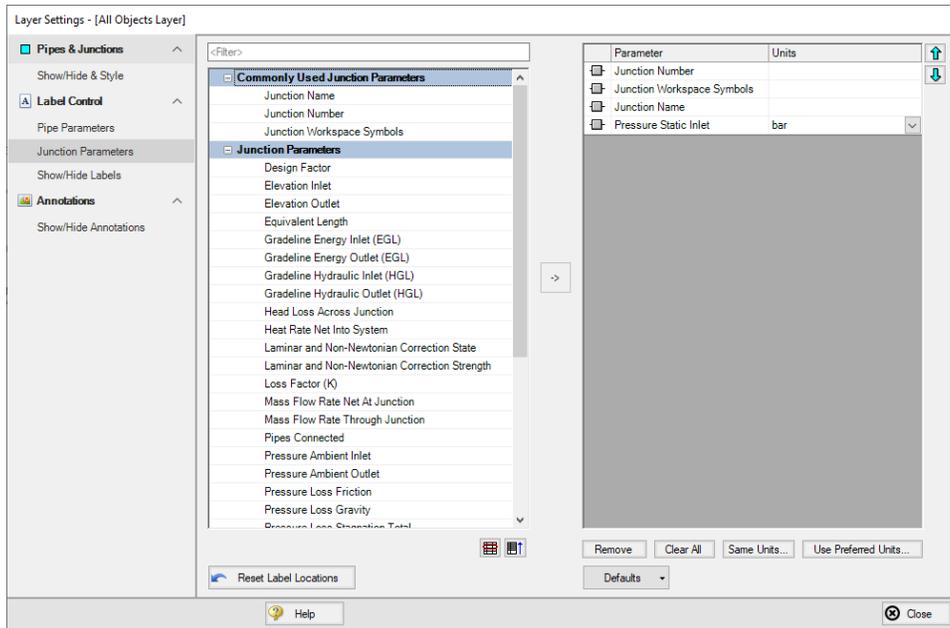


Figure 21: The Layer Settings window selects visible parameters for the Workspace Layers

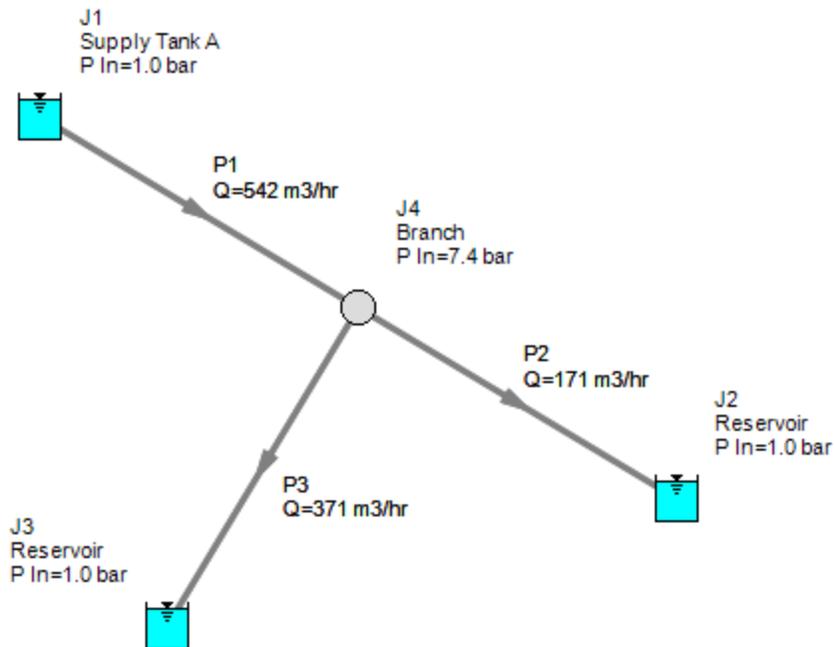


Figure 22: The Workspace window displays input and output data utilizing Workspace Layers

D. Graph the results

➤ **Change to the Graph Results window** by clicking the Graph Results tab. The Graph Results window offers full-featured Windows plot preparation.

The Graph Parameters menu will automatically be displayed in the Quick Access Panel on the far right of the Graph Results Window and is where you are to specify which graphs to generate.

The Profile graph tab in the Graph Parameters menu will also be selected by default on Quick Access Panel. Make sure the Plot Single Path option is selected from the Pipes drop-down menu. Select pipes 1 and 2 from the pipes list. Choose Pressure Static from the Parameter drop-down menu. Make sure meters is selected as the Length Units and bar as the Pressure Static units (see Figure 23). Then click Generate. The Show Junction Locations feature is located on the Graph Results toolbar and can be turned on by clicking the icon.

A graph appears showing the static pressure along the flow-path defined by pipes 1 and 2 in Figure 24.

You can use the other buttons in the Graph Results window to change the graph appearance and to save and import data for cross-plotting. The Graph Results window can be printed, saved to file, copied to the clipboard, or printed to an Adobe PDF file. The graph's x-y data can be exported to file or copied to the clipboard.

Note that the graph guide, located at the top right of the Graph Results window and represented with the What Would You Like to Do? icon, can guide you through the development of your graph. This feature can be hidden by clicking on the icon.

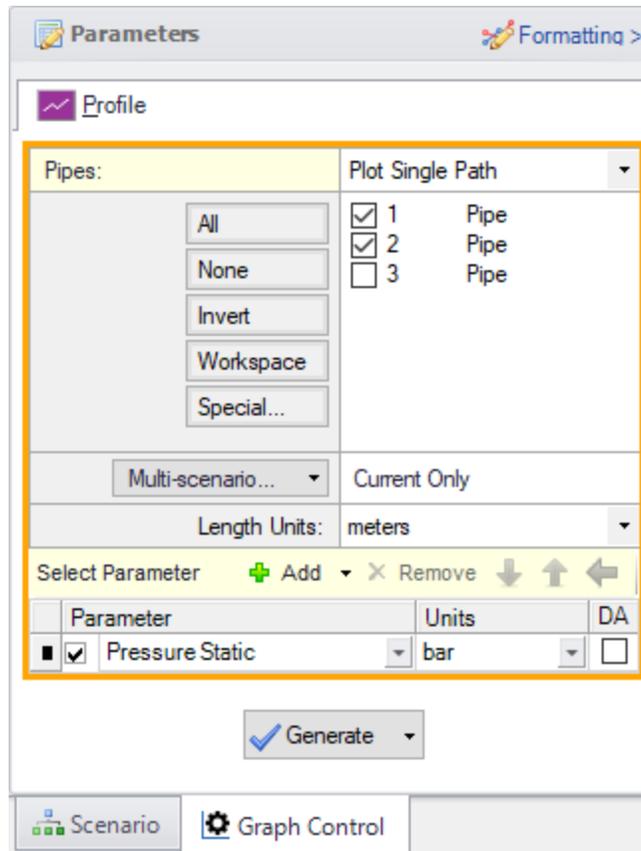


Figure 23: The Graph Control window controls the Graph Results content

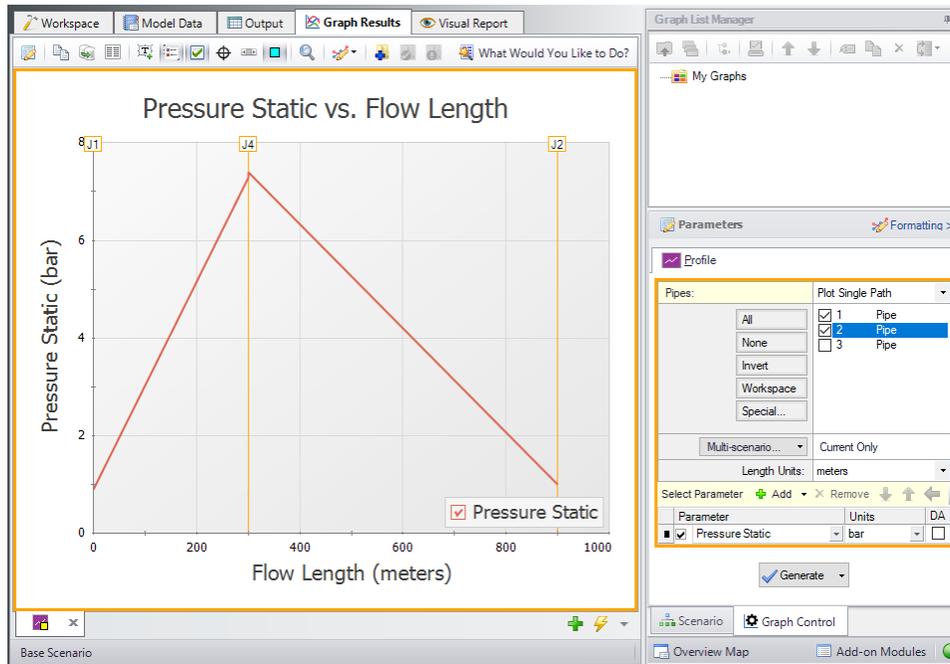


Figure 24: The Graph Results window offers full-featured plot generation

Conclusion

You have now used AFT Fathom's Primary Windows and functions to build a simple model. Review the rest of the Fathom Help File contents for more detailed information on each of the windows and functions.

Sizing a Pump

Summary

This example will walk you through a simple calculation to size a pump.

A pump is to be used to transfer water from a supply reservoir to a holding reservoir at the top of a hill. The system consists of a supply reservoir, a pump, a discharge reservoir, and two pipes.

The first pipe from the supply reservoir to the pump is 3 meters long, and the second pipe leading from the pump to the discharge reservoir is 300 meters long. Both pipes are 4 inch, Steel - ANSI, STD (Schedule 40) pipes. Constant fluid properties are assumed.

The supply reservoir has a liquid surface pressure of 0 barG (0 kPa (g)), and the liquid surface is at an elevation of 3 meters. The discharge reservoir has a liquid surface pressure of 0 barG (0 kPa (g)), and the liquid surface is at an elevation of 60 meters. The pipes for both reservoirs connect to the reservoirs at a depth of 3 meters.

What is the head requirement for a pump to supply flow in this system at 115 m³/hr?

Topics Covered

- Specifying a pump in the Pump Properties Window
- Defining pipe fittings and losses lumped into a pipe
- Sizing a pump
- Entering pump curve data

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Sizing a Pump.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

Sizing a Pump

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = AFT Standard
 - b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
 - c. Temperature = 21 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

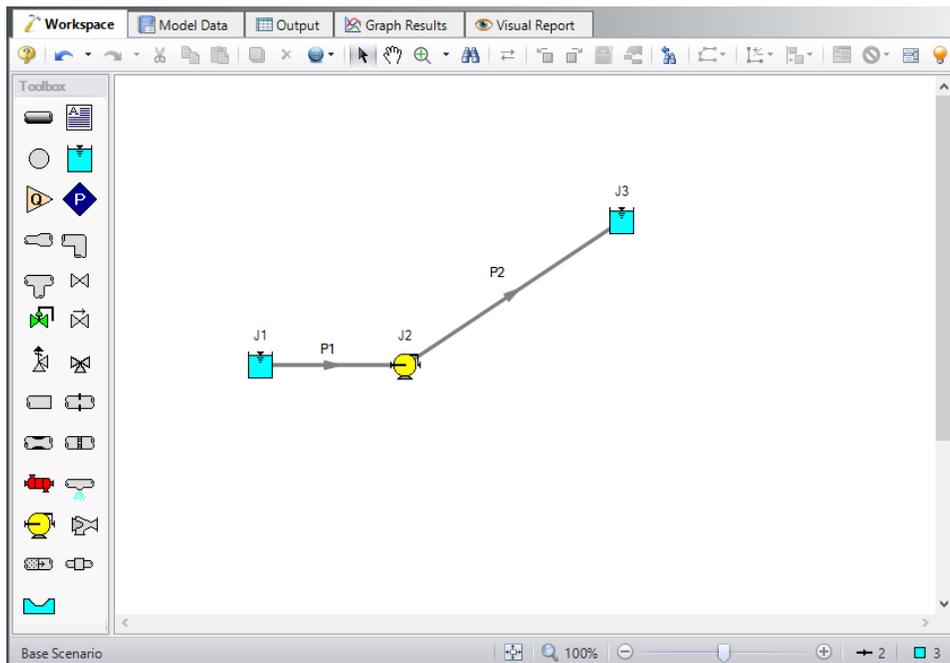


Figure 1: Workspace for Sizing a Pump Example

The system is in place but now we need to enter the properties of the objects. Double-click each pipe and junction and enter the following properties. The required information is highlighted in blue.

Pipe Properties

1. Pipe Model tab
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 4 inch
 - d. Type = STD (schedule 40)
 - e. Friction Model Data Set = Standard
 - f. Lengths =

Pipe	Length (meters)
1	3
2	300

The pipe model also allows for fittings and losses like valves or elbows. Select the Fittings & Losses tab for P2, and type in 25 for the Total K Factor. Now click OK to close the Pipe Properties window and accept your changes.

Note: A red ampersand (&) will now appear next to the pipe number for pipe P2. This indicates that Fittings & Losses are entered in the Pipe Properties window. This symbol can be edited or hidden in User Options.

Junction Properties

1. J1 Reservoir
 - a. Name = Lower Reservoir
 - b. Liquid Surface Elevation = 3 meters
 - c. Liquid Surface Pressure = 0 barG (0 kPa (g))
 - d. Pipe Depth = 3 meters
2. J2 Pump
 - a. Inlet Elevation = 0 meters
 - b. Pump Model = Centrifugal (Rotodynamic)
 - c. Analysis Type = Sizing
 - d. Parameter = Volumetric Flow Rate
 - e. Fixed Flow Rate = 115 m³/hr
3. J3 Reservoir
 - a. Name = Upper Reservoir
 - b. Liquid Surface Elevation = 60 meters
 - c. Liquid Surface Pressure = 0 barG (0 kPa (g))
 - d. Pipe Depth = 3 meters

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will

Sizing a Pump

have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

The Output window (Figure 2) contains all the data that was specified in the output control window. Because we were interested in the pump requirements for this system, select the Pump Summary tab in the General (top) window. The pump data can also be found in the Junction (bottom) window. We are primarily interested in the pump head (or pressure) requirement. This information can be found in the Pump Summary window.

Select Output Control from the Common Toolbar or Tools menu to open the Output Control window. Click the General tab, enter the title Water Transfer System, then click OK to save the title.

For this system, the head requirement of the pump is 116.3 meters. Because the difference in elevation of the reservoirs was 57 meters this is the minimum head rise regardless of flow rate or pipe size. Coincidentally, the friction loss was also approximately 59 meters. This can be seen in the dH column of the Pipe output window.

The screenshot shows the AFT Fathom Output window with the following data tables:

Jct	Results Diagram	Name	Vol. Flow (m ³ /hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m ³ /hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
2	Show	Pump	115.0	31.90	11.39	116.3	100.0	N/A	36.38	N/A	N/A	12.69	N/A

Pipe	Name	Vol. Flow Rate (m ³ /hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)
1	Pipe	115.0	3.889	1.232	1.193	0	0.00	0.03885	0.03885	0.000	0.3966	1.232	1.193	1.307	1.268
2	Pipe	115.0	3.889	12.587	1.232	0	57.00	11.35588	11.35588	5.583	58.9460	12.587	1.232	12.663	1.307

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m ³ /hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Lower Reservoir	1.013	1.307	1.013	1.307	115.0	31.90	0
2	Pump	1.193	12.587	1.268	12.663	115.0	31.90	0
3	Upper Reservoir	1.013	1.307	1.013	1.307	115.0	31.90	0

Figure 2: Output window for Pump Sizing Example

Step 6. Add a Pump Curve

Now that the pump head requirement has been identified as 116.3 meters at the design flow rate of 115 m³/hr, a pump of the correct size can be purchased to meet this requirement. Once the actual pump has been identified, the pump characteristics can be added to the pump by adding the pump curve data to the pump junction.

Open the Pump Properties window and enter the following data:

1. Pump Model = Centrifugal (Rotodynamic)
2. Analysis Type = Pump Curve
3. Enter Curve Data
 - a. Flow Q Parameter = Volumetric
 - b. Head/Pressure dH Parameter = Head

Volumetric	Head
m ³ /hr	meters
0	122
115	116
230	76

- c. Curve Fit Order = 2
- d. Click Generate Curve Fit Now
- e. Click OK

Note: This is a situation where a user could create a new scenario using the Scenario Manager to examine a what-if situation, without disturbing the basic model. For an example that illustrates how to use the Scenario Manager, see the Water to Housing Project example.

Sizing a Pump

Pump Configuration

Configuration Method
 Simple Multiple

Pump Data Configuration Data Performance Graph

Raw Data: Optional Data

Parameter	Flow Q	Head/Pressure dH	NPSHR	Efficiency/Power
Units	Volumetric	Head		Efficiency
	m3/hr	meters	meters	Percent
1	0	122		
2	115	116		
3	230	76		
4				
5				
6				
7				
8				
9				
10				
11				
12				
13				
14				

Curve Fitting
 Curve Fit Type:
 Polynomial
 Interpolated X-Y Data

Head Rise
 NPSHR
 Efficiency

All None Invert

Curve Fit Order
 2

Generate Curve Fit Now

PUMP CURVE

a	122
b	0.09565217
c	-0.001285444
NPSHR	None
EFFICIENCY	None

$\Delta H = a + bQ + cQ^2 + dQ^3 + eQ^4$

Reference Density ?
 No Correction Water @STP
 User Specified kg/m3

OK Cancel Help

Figure 3: The Pump Configuration Window is used to enter pump curve data

Step 7. Re-evaluate the model

Re-run this model. Examine the Pump Summary in the General Output Section. The output (Figure 4) shows the pump operating at a flow rate of 114.7 m3/hr, and a head rise of 116.1 meters, which is acceptably close to the previous sizing calculation.

Sizing a Pump

Jct	Results Diagram	Name	Vol. Flow (m3/hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
2	Show	Pump	114.7	31.82	11.37	116.1	N/A	100.0	36.19	N/A	N/A	12.70	N/A

Pipe	Name	Vol. Flow Rate (m3/hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)
1	Pipe	114.7	3.879	1.232	1.193	0	0.00	0.03864	0.03864	0.000	0.3945	1.232	1.193	1.307	1.268
2	Pipe	114.7	3.879	12.561	1.232	0	57.00	11.32874	11.32874	5.583	58.6689	12.561	1.232	12.636	1.307

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Lower Reservoir	1.013	1.307	1.013	1.307	114.7	31.82	0
2	Pump	1.193	12.561	1.268	12.636	114.7	31.82	0
3	Upper Reservoir	1.013	1.307	1.013	1.307	114.7	31.82	0

Figure 4: Output when running with a pump curve

Change to the Graph Results window (e.g., using the Windows menu), and click on the Pump vs. System tab (Figure 5) in the Graph Parameters menu on the Quick Access Panel. Click Generate to graph a pump vs. system curve, as shown in Figure 6.

Parameters
Formatting >

Pump vs. System

Profile

Curve Type:	Single										
Flow Type:	Volumetric m3/hr										
Pressure or Head:	Head meters										
Pump	<table border="1" style="width: 100%; border-collapse: collapse;"> <thead> <tr> <th style="padding: 2px 5px;">CRV</th> <th style="padding: 2px 5px;">FUL</th> <th style="padding: 2px 5px;">MIN</th> <th style="padding: 2px 5px;">AOR</th> <th style="padding: 2px 5px;">POR</th> </tr> </thead> <tbody> <tr> <td style="padding: 2px 5px;"><input checked="" type="checkbox"/></td> <td style="padding: 2px 5px;"></td> <td style="padding: 2px 5px;"></td> <td style="padding: 2px 5px;"></td> <td style="padding: 2px 5px;"></td> </tr> </tbody> </table>	CRV	FUL	MIN	AOR	POR	<input checked="" type="checkbox"/>				
CRV	FUL	MIN	AOR	POR							
<input checked="" type="checkbox"/>											
Flow Rate Range:	<input checked="" type="radio"/> Auto <input type="radio"/> User Specified										
User Range:	0 138 m3/hr										
Data Points Per Curve:	30										

Generate

Scenario
Graph Control

Figure 5: Select Graph Data for creating a pump vs. system curve

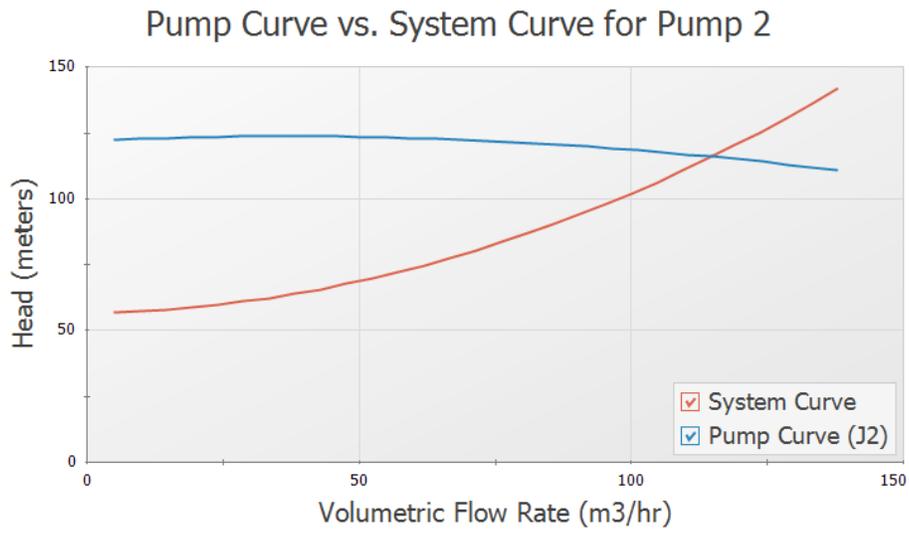


Figure 6: Pump vs. system curve using selected pump

Conclusion

AFT Fathom can be used to size pumps. Once the actual pump has been selected, the model can be modified to include the actual pump curve data. Then the actual system properties and behavior can be calculated.

Control Valve

Summary

This will walk you through a simple calculation to calculate a pipe size for a control valve system. Water will be flowing from a reservoir on a hill down to another reservoir. The pipe must be sized correctly to allow 57 m³/hr to flow from one reservoir to the other. A control valve is used to control this flow, but the valve must have a minimum pressure drop of 0.65 bar (65 kPa) to operate correctly. Only STD (schedule 40) pipes will be considered.

Topics Covered

- Using flow control valves
- Entering and changing pipe size data

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Control Valve.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

- a. Fluid Library = AFT Standard
- b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
- c. Temperature = 21 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

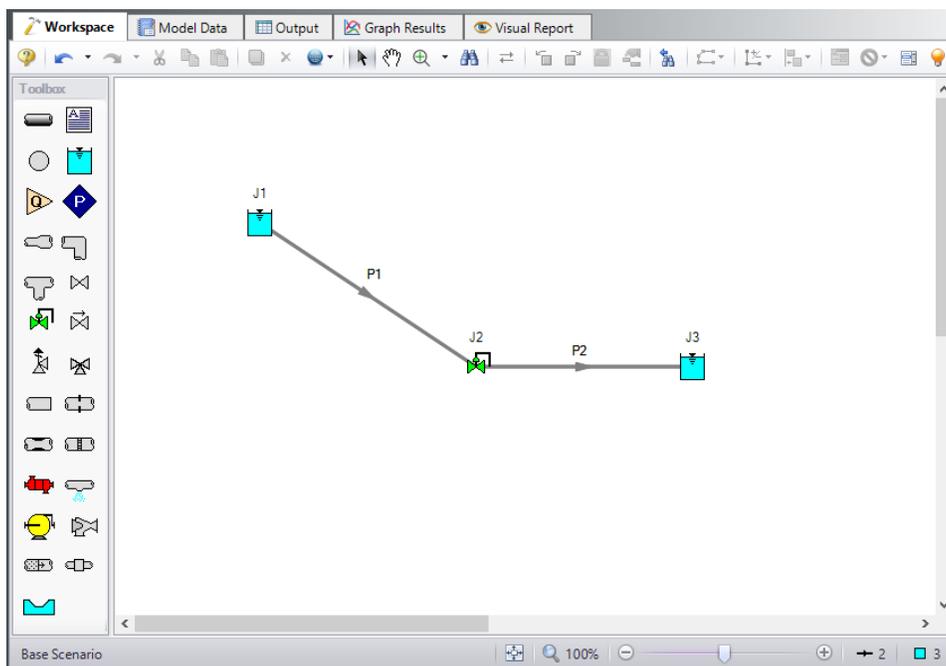


Figure 1: Workspace for Control Valve Problem

The system is in place but now we need to enter the properties of the objects. For this example, we need to find the correct pipe size but we are limited to STD (schedule 40) pipe sizes. We need to use our engineering judgment to take an initial guess at the pipe size for 57 m³/hr. Let's try 3 inch pipe. Specify the pipes as the following:

Pipe Properties

1. All Pipes
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 3 inch

- d. Type = STD (schedule 40)
- e. Friction Model Data Set = Standard
- f. Length = 30 meters

Junction Properties

- 1. J1 Reservoir
 - a. Name = Upper Reservoir
 - b. Liquid Surface Elevation = 12 meters
 - c. Liquid Surface Pressure = 0 barG
 - d. Pipe Depth = 0 meters
- 2. J2 Control Valve
 - a. Inlet Elevation = 3 meters
 - b. Valve Type = Flow Control (FCV)
 - c. Control Setpoint = Volumetric Flow Rate
 - d. Flow Setpoint = 57 m³/hr
- 3. J3 Reservoir
 - a. Name = Lower Reservoir
 - b. Liquid Surface Elevation = 3 meters
 - c. Liquid Surface Pressure = 0 barG
 - d. Pipe Depth = 0 meters

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

Open the Output Control window by selecting Output Control from the Output toolbar or Tools menu. This window allows the user to specify the output from the Fathom simulation. The window starts with a default set of output parameters which you are free to change.

In this example, we want to review the effects of changing the nominal pipe size. To simplify this, let's add this parameter to the output.

- 1. From the Pipes output list, select the parameter Nominal Size in the list on the left.
- 2. Click Add (or double-click the parameter), which adds the parameter to the list on the right.

3. Use the Reorder buttons (in the upper right) to move the Nominal Size up so that it becomes the second parameter in the list (see Figure 2).
4. Click OK.

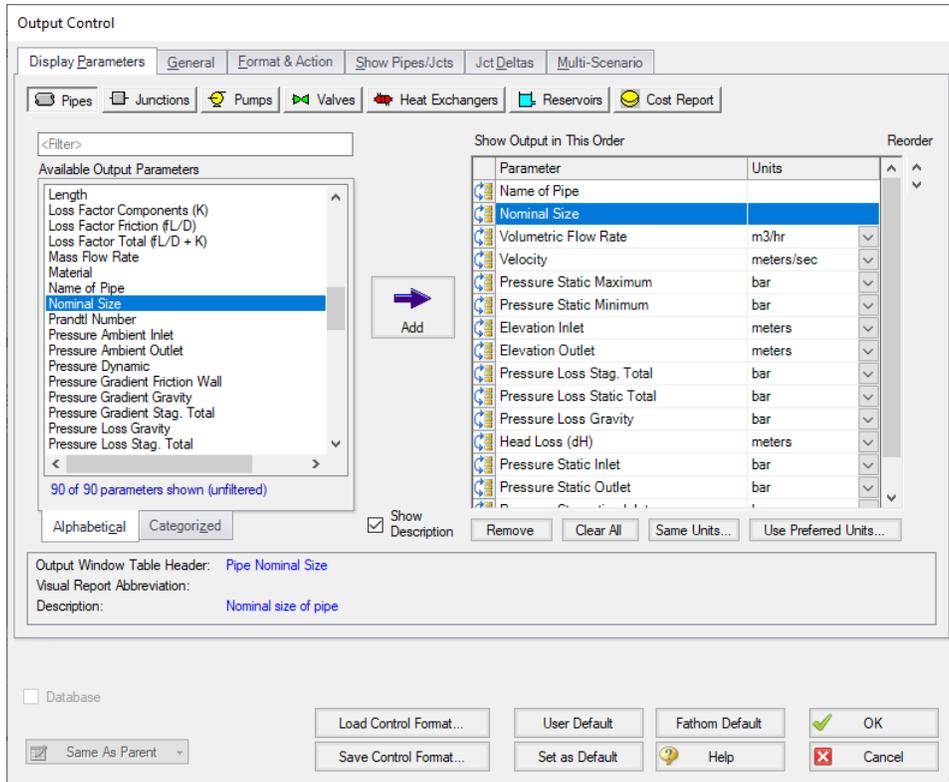


Figure 2: Output Control Window

The Output window contains all the data that was specified in the output control window. The output is shown in Figure 3. In the summary window, there is a Valve Summary tab. Select this tab. You can see that the flow control valve achieved the flow rate of 57 m3/hr but the pressure drop (dP) is only 0.08327 bar (8.327 kPa) (see Figure 3). The pressure drop requirement is 0.65 bar (65 kPa) minimum. This pressure drop is too low. The pipe size needs to be increased.

Control Valve

Jct	Name	Valve Type	Vol. Flow Rate (m3/hr)	Mass Flow (kg/sec)	dP Stag. (bar)	dH (meters)	P Static In (bar)	Cv	K	Valve State
2	Control Valve	FCV	57.00	15.81	0.08327	0.8502	1.441	228.3	1.513	Open

Pipe	Name	Pipe Nominal Size	Vol. Flow Rate (m3/hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)
1	Pipe	3 inch	57.00	3.320	1.441	0.9582	12.000	3.000	-0.4824	-0.4824	-0.8815	4.075	0.9582	1.4406	1.013	1.496
2	Pipe	3 inch	57.00	3.320	1.357	0.9582	3.000	3.000	0.3991	0.3991	0.0000	4.075	1.3573	0.9582	1.412	1.013

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Upper Reservoir	1.013	1.013	1.013	1.013	57.00	15.81	0.000
2	Control Valve	1.441	1.357	1.496	1.412	57.00	15.81	1.513
3	Lower Reservoir	1.013	1.013	1.013	1.013	57.00	15.81	0.000

Figure 3: Output Window for Control Valve Example

Step 6. Change Pipe Sizes

Go back to the Workspace and change the pipe properties to a new pipe size. By trial and error, you can determine that by using 4 inch for both pipes, a pressure drop of 0.6797 bar (67.97 kPa) can be obtained (see Figure 4).

Jct	Name	Valve Type	Vol. Flow Rate (m3/hr)	Mass Flow (kg/sec)	dP Stag. (bar)	dH (meters)	P Static In (bar)	Cv	K	Valve State
2	Control Valve	FCV	57.00	15.81	0.6797	6.939	1.775	79.92	36.62	Open

Pipe	Name	Pipe Nominal Size	Vol. Flow Rate (m3/hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)
1	Pipe	4 inch	57.00	1.928	1.775	0.9947	12.000	3.000	-0.7806	-0.7806	-0.8815	1.030	0.9947	1.7753	1.013	1.794
2	Pipe	4 inch	57.00	1.928	1.096	0.9947	3.000	3.000	0.1009	0.1009	0.0000	1.030	1.0956	0.9947	1.114	1.013

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Upper Reservoir	1.013	1.013	1.013	1.013	57.00	15.81	0.00
2	Control Valve	1.775	1.096	1.794	1.114	57.00	15.81	36.62
3	Lower Reservoir	1.013	1.013	1.013	1.013	57.00	15.81	0.00

Figure 4: Output Window for modified Control Valve Example

Conclusion

To meet the system requirements, use 4-inch pipe.

Heat Transfer in a Pipe

Summary

The objective of this example is to demonstrate how to calculate heat transfer in a pipe. For this example we will specify a mass flow rate and an inlet temperature and calculate the outlet temperature and total heat loss for a 30 meter section of pipe.

Topics Covered

- Modeling heat transfer in a single pipe

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Heat Transfer in a Pipe.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = AFT Standard
 - b. Fluid = Water (liquid)

- i. After selecting, click Add to Model
 - c. Temperature = 21 deg. C
 - d. This calculates the default fluid properties to use in the model to initialize the heat transfer calculations.
3. Open the Heat Transfer/Variable Fluids panel:
 - a. Heat Transfer = Heat Transfer With Energy Balance (Single Fluid)

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

Although one pipe is all that we are modeling, we still need junctions at each end to define the pressure and flow conditions for this pipe.

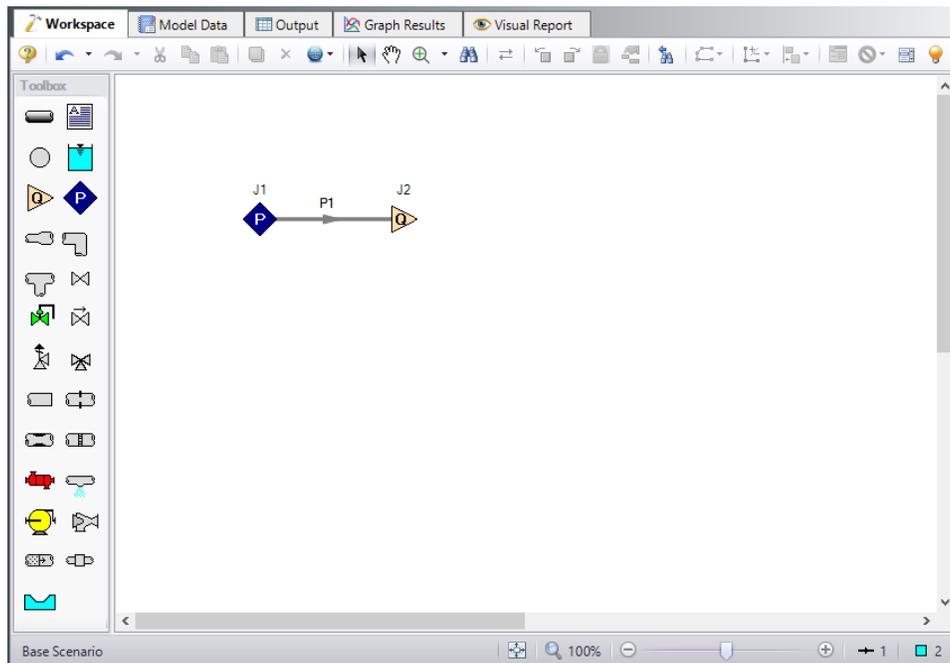


Figure 1: Workspace for Heat Transfer Problem

The system is in place but now we need to enter the properties of the objects. Double-click each pipe and junction and enter the following properties. The required information is highlighted in blue.

Pipe Properties

Because heat transfer was selected on the Heat Transfer/Variable Fluids panel in Analysis Setup, the Pipe Properties window has an additional tab to define: Heat Transfer. We will use Convective Heat

Transfer for this example. Fathom allows you to perform a layered heat transfer analysis with internal and external convection coefficients and conduction through multiple insulation layers. For this example, we will consider the internal convection, conduction through the pipe wall, and external convection as affecting heat transfer in our system.

Pipe P1

1. Pipe Model tab
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 6 inch
 - b. Type = STD (schedule 40)
 - c. Friction Model Data Set = Standard
 - d. Length = 100 meters
2. Heat Transfer tab
 - a. Heat Transfer Model = Convective Heat Transfer
 - b. Ambient Conditions
 - i. Temperature = 0 deg. C
 - ii. Fluid Velocity = 30 km/hr
 - iii. Fluid = Air
 - c. Fluid Internal
 - i. Apply = Checked
 - ii. Conductivity/Convection Data Source = Correlation
 - iii. HT Area Ratio = 1
 - iv. Convection Coefficient = Gnielinski
 - d. Pipe Wall
 - i. Apply = Checked
 - ii. Conductivity/Convection Data Source = Material Library
 - iii. Conductivity = From Library
 - iv. Thickness = 0.7111999 cm
 - v. HT Area Ratio = 1
 - e. External
 - i. Apply = Checked
 - ii. Conductivity/Convection Data Source = Correlation
 - iii. HT Area Ratio = 1
 - iv. Convection Coefficient = Forced (Churchill-Bernstein)

The heat transfer window looks like Figure 2 below.

Heat Transfer in a Pipe

Pipe Properties

Number: 1 Upstream Junction: 1
Name: Pipe Downstream Junction: 2
Copy Data From Pipe... Copy Previous...

OK
Cancel
Jump...
Help

Pipe Model | Fittings & Losses | **Heat Transfer** | Design Alerts | Optional | Notes | Status

Heat Transfer Model: Convective Heat Transfer
Resistance Geometry: Radial
Number of Insulation Layers: None

Ambient Conditions
Temperature: 0 deg. C
Fluid Velocity: 30 km/hr
Fluid: Air Water

Load Default

Resistance Type	Apply	Conductivity/Convection Data Source	Conductivity	Thickness	HT Area Ratio	Convection Coefficient
Units			W/m-K	cm		W/m2-K
Fluid Internal	<input checked="" type="checkbox"/>	Correlation			1	Gnielinski
Pipe Wall	<input checked="" type="checkbox"/>	Material Library	From Library	0.7111999	1	
External	<input checked="" type="checkbox"/>	Correlation			1	Forced (Churchill-Bernstein)

Figure 2: Pipe properties window for Heat Transfer Problem

Junction Properties

Now that the pipe is defined, specify the junction properties:

1. J1 Assigned Pressure
 - a. Elevation = 0 meters
 - b. Pressure = 3.45 bar (345 kPa)
 - c. Temperature = 65 deg. C
 - d. Pressure Specification = Stagnation
2. J2 Assigned Flow
 - a. Elevation = 0 meters
 - b. Type = Outflow
 - c. Flow Specification = Mass Flow Rate
 - d. Flow Rate = 4.5 kg/sec

Note: The temperature at J2 will not be used by Fathom because the outlet temperature of the pipe is defined by the inlet temperature pipe due to the Assigned Pressure junction and the heat loss through the pipe.

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

Open the Output Control window by selecting Output Control from the Output toolbar or Tools menu. From the Display Parameters tab, Pipes selection, double-click the parameters Heat Flux, Heat Rate, Temperature Inlet, and Temperature Outlet in the left list to add these output parameters to this model. Click OK to return to the Output window.

The Output window contains all the data that was specified in the output control window. Because we added Inlet and Outlet temperatures to the output, we can see that the outlet temperature is 59.70 deg. C. The output is shown in Figure 3 below.

Note: If the default Output Control parameters are being displayed, a warning will be shown stating the *HGL, EGL, and head loss results may not be meaningful for variable density systems* as can be seen in Figure 3. If all head output parameters are hidden using Output Control, this warning will no longer appear.

The screenshot shows the 'Output' window with the following data tables:

Pipe	Name	Vol. Flow Rate (m ³ /hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag In (bar)	P Stag Out (bar)	Heat Flux (kW/m ²)	Heat Rate (kW)	T Inlet (deg. C)	T Outlet (deg. C)
1	Pipe	16.48	0.2456	3.450	3.446	0	0	3.860E-03	3.860E-03	0	0.04004	3.450	3.446	3.450	3.446	-2.059	-99.64	65.00	59.70

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag In (bar)	P Stag Out (bar)	Vol. Flow Rate Thru Jct (m ³ /hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Assigned Pressure	3.450	3.450	3.450	3.450	16.48	4.500	0
2	Assigned Flow	3.446	3.446	3.446	3.446	16.48	4.500	0

Figure 3: Output window for Heat Transfer Problem

Heat Exchanger System

Summary

This example will walk you through a simple calculation to size a pump for a closed loop heat exchanger system. First you will build the system with a fixed flow pump and then enter a pump curve that meets these requirements.

Topics Covered

- Sizing a pump for a specific system
- Entering a loss curve for a Heat exchanger
- Graphing system curves versus pump curves

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Heat Exchanger System.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

- a. Fluid Library = AFT Standard
- b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
- c. Temperature = 21 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

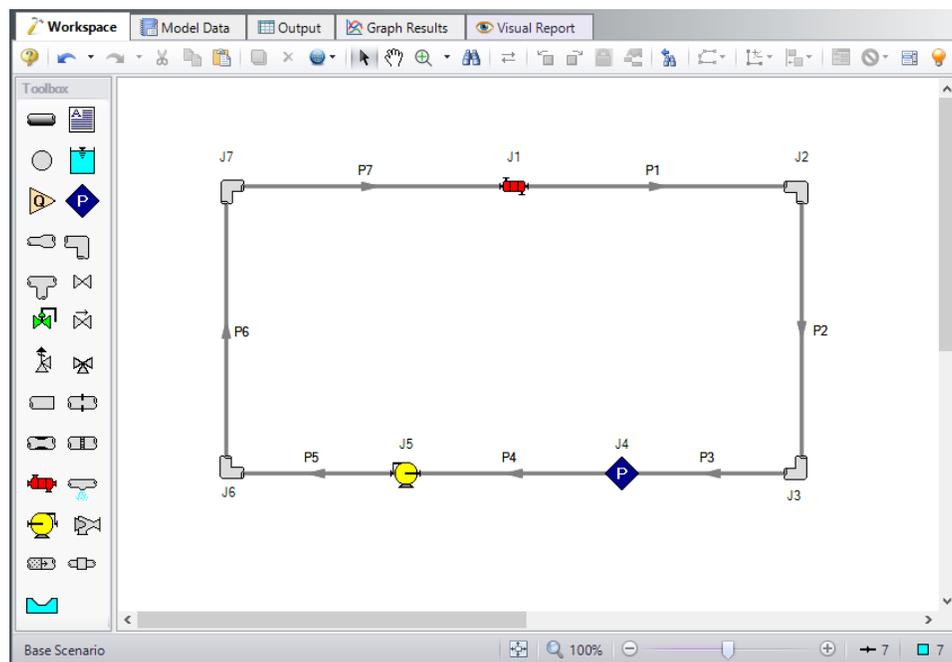


Figure 1: Layout of Heat Exchanger Problem

The system is in place but now we need to enter the properties of the objects. Double-click each pipe and junction and enter the following properties. The required information is highlighted in blue.

Pipe Properties

1. Pipe Model tab
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 4 inch
 - d. Type = STD (schedule 40)

- e. Friction Model Data Set = Standard
- f. Lengths =

Pipe	Length (meters)
1	60
2	5
3	30
4	30
5	60
6	5
7	60

Junction Properties

Specify each junction by entering the following properties in each corresponding Junction Properties window:

- 1. J1 Heat Exchanger
 - a. Inlet Elevation = 5 meters
 - b. Loss Model = Resistance Curve
 - c. Enter Curve Data =

Volumetric	Pressure
m ³ /hr	bar
0	0
25	0.35
50	1.4

- d. Curve Fit Order = 2
 - e. Click Generate Curve Fit Now
- 1. J2 & J7 Bend
 - a. Inlet Elevation = 5 meters
 - b. Type = Standard Elbow (knee, threaded)
 - c. Angle = 90 Degrees
- 2. J3 & J6 Bend
 - a. Inlet Elevation = 0 meters
 - b. Type = Standard Elbow (knee, threaded)
 - c. Angle = 90 Degrees

3. J4 Assigned Pressure
 - a. Elevation = 0 meters
 - b. Pressure = 7 bar (700 kPa)
 - c. Pressure Specification = Stagnation
4. J5 Pump
 - a. Inlet Elevation = 0 meters
 - b. Pump Model = Centrifugal (Rotodynamic)
 - c. Analysis Type = Sizing
 - d. Parameter = Volumetric Flow Rate
 - e. Fixed Flow Rate = 25 m3/hr

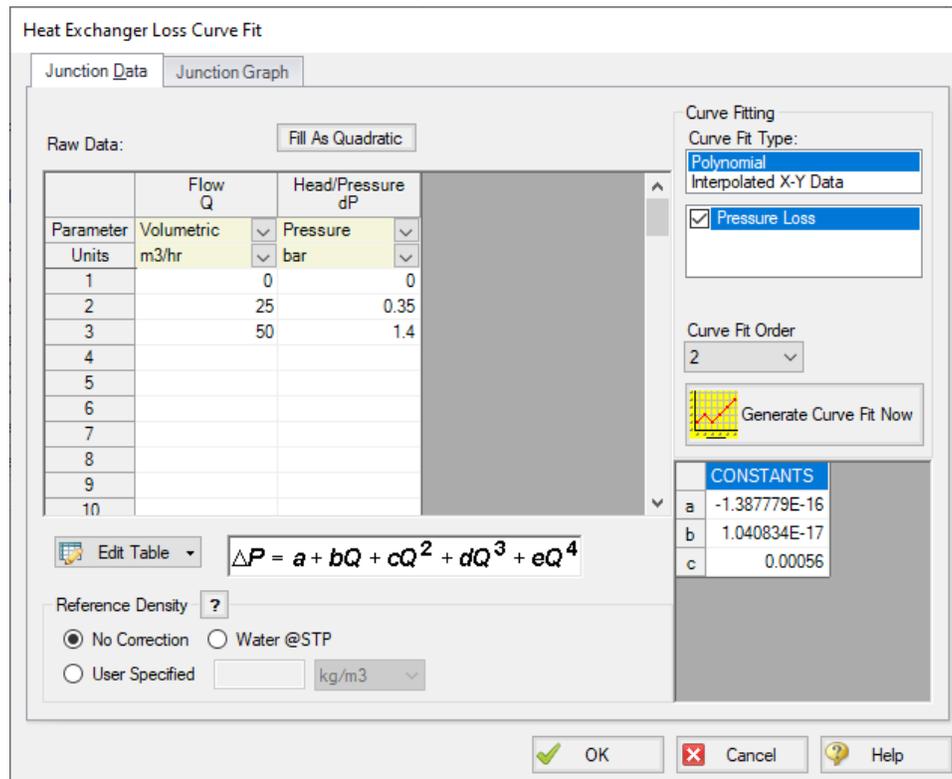


Figure 2: Heat Exchanger Loss Curve Fit Specification

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the

results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

The Output window contains all the data that was specified in the output control window. Because we are interested in the pump requirements for this system, select the Pump Summary tab in the General (top) window. The pump data can also be found in the Junction window (bottom). If we are primarily interested in the head (or pressure) requirement, this is found in the summary window.

For this system, the head requirement of the pump is 0.5359 bar (53.59 kPa) (5.472 meters). At the assigned flow rate of 25.00 m³/hr, the pressure drop across the heat exchanger was 0.3500 bar (35.00 kPa) (3.574 meters) so the total pipe and elbow loss was approximately 0.1859 bar (18.59 kPa) (1.898 meters). This output is shown in Figure 3a and Figure 3b.

Jct	Results Diagram	Name	Vol. Flow (m ³ /hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m ³ /hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
5	Show	Pump	25.00	6.936	0.5359	5.472	100.0	N/A	0.3719	N/A	N/A	71.00	N/A

Figure 3a: Pump Summary for Heat Exchanger Example

Jct	Name	Vol. Flow (m ³ /hr)	Mass Flow (kg/sec)	dP Stag. (bar)	dH (meters)	dT Loss (deg. C)	Heat Rate In (kW)	T Inlet (deg. C)	T Outlet (deg. C)	T 2nd Inlet (deg. C)	T 2nd Outlet (deg. C)
1	Heat Exchanger	25.00	6.936	0.3500	3.574	N/A	N/A	21.00	21.00	N/A	N/A

Figure 3b: Heat Exchanger Summary for Heat Exchanger Example

Step 6. Add Pump Curve

Now that we know the general requirements of the pump, we can select an appropriate pump and enter this data. Open the Pump Properties window and enter the following data:

1. Pump Model = Centrifugal (Rotodynamic)
2. Analysis Type = Pump Curve
3. Enter Curve Data
 - a. Flow Q Parameter = Volumetric
 - b. Head/Pressure dH Parameter = Head

Heat Exchanger System

Volumetric	Head
m3/hr	meters
0	6
25	5.5
50	3.5

- c. Curve Fit Order = 2
- d. Click Generate Curve Fit Now

Step 7. Re-evaluate the Model

Select Run Model from the toolbar or Analysis menu. Because the pump head at 25 m3/hr was slightly higher than required, the system will adjust with a slightly higher flow rate. The output of this run is shown in Figure 4.

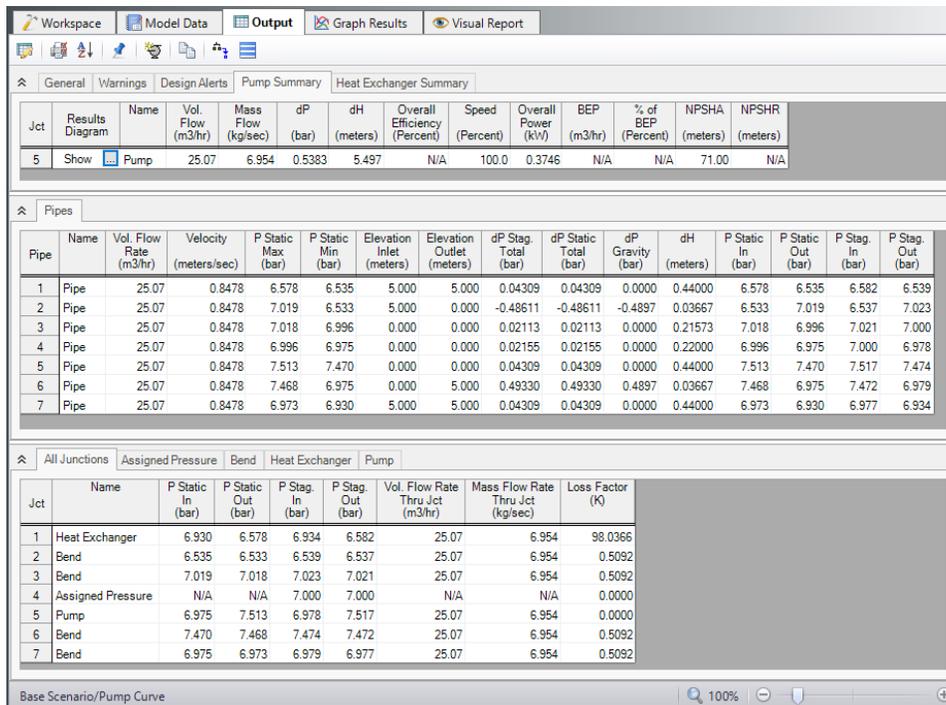


Figure 4: Output window for Heat Exchanger Example with real pump data

This system is a good example to demonstrate how to use the Graph Results Tab. Change to the Graph Results window by choosing it from the Window drop-down on the menu bar, clicking the Graph Results tab on the toolbar, or by pressing CTRL+G. Now the Graph Control window will be displayed on the Quick Access Panel as shown in Figure 5. Select Pump Curve vs. System Curve and Fathom automatically selects the data you want to graph.

Parameters
 Formatting >

Pump vs. System
 Profile

Curve Type:	Single				
Flow Type:	Volumetric	m3/hr			
Pressure or Head:	Head	meters			
Pump	CRV	FUL	MIN	AOR	POR
5 Pump (25.0675 m3/hr)	<input checked="" type="checkbox"/>				
Flow Rate Range:	<input checked="" type="radio"/> Auto <input type="radio"/> User Specified				
User Range:	0	30.1	m3/hr		
Data Points Per Curve:	30				

Scenario
 Graph Control

Figure 5: Select Graph Data window for Heat Exchanger System

Click Generate to view this graph. This graph is shown in Figure 6. The red line shows the pump curve that we entered. This starts out with 6 meters of head available at 0 m3/hr and decreases slowly with increasing flow rate. The blue line is the overall system curve. The head requirement increases significantly with flow rate. Most of the pressure loss (head loss) is through the heat exchanger but this graph also includes the pipe and elbow loss. The lines cross right at 25.06 m3/hr which is the operating point shown in the output in Figure 4.

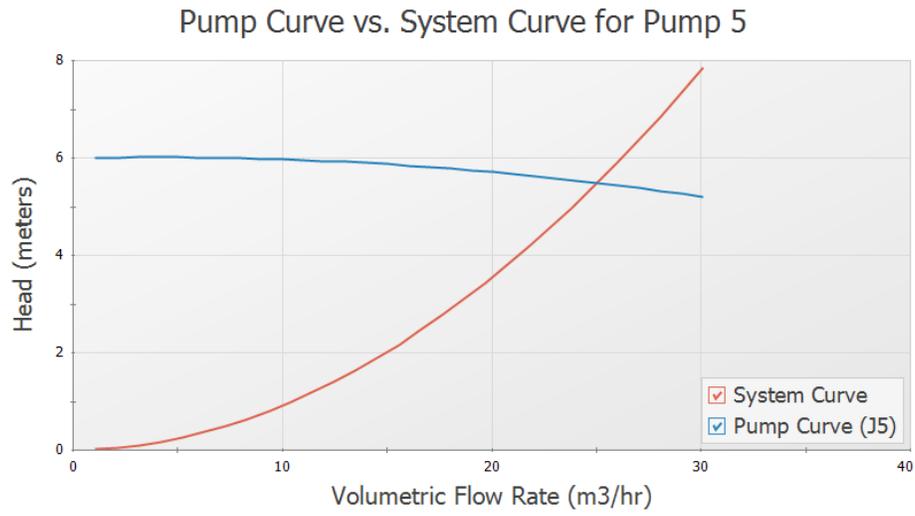


Figure 6: Pump Curve vs. System Curve for Heat Exchanger System

Spray Discharge System

Summary

The objective of this example is to find the minimum supply pressure needed to supply a system of eight spray discharge heads with at least 25 m³/hr to each spray head. The system has one main supply line that branches into two supply lines that supply four sprayers each.

Topics Covered

- Using spray discharge junctions
- Iterating on a supply condition to reach a desired output condition
- Using Isometric Pipe Drawing Mode

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- *Metric - Spray Discharge System.fth*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

Spray Discharge System

- a. Fluid Library = AFT Standard
- b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
- c. Temperature = 21 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

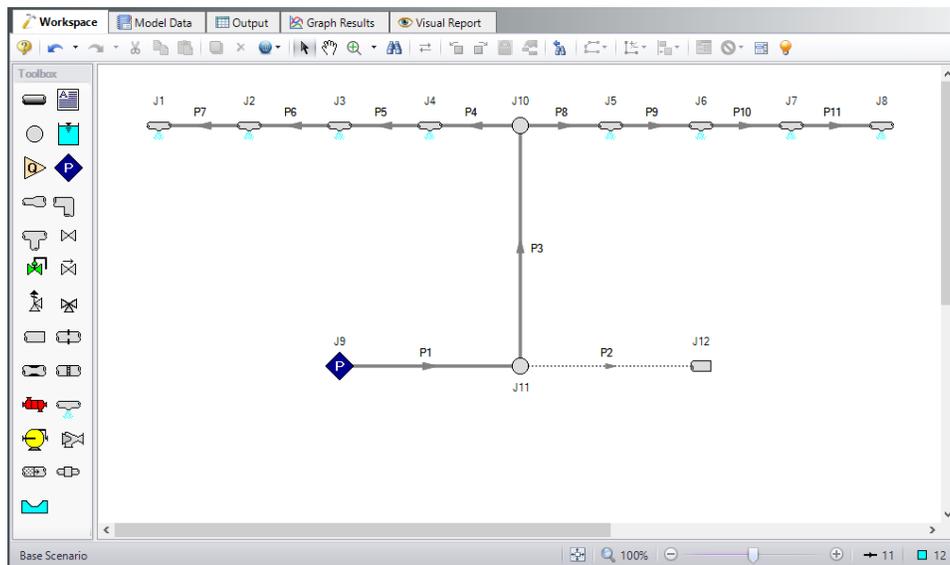


Figure 1: Workspace for Spray Discharge System

The system is in place but now we need to enter the properties of the objects. Double-click each pipe and junction and enter the following properties. The required information is highlighted in blue.

Pipe Properties

1. Pipe Model tab
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = Use table below
 - d. Type = STD (schedule 40)
 - e. Friction Model Data Set = Standard
 - f. Lengths =

Spray Discharge System

Pipe	Size	Length (meters)
1 & 2	8 inch	150
3	4 inch	3
4 - 11	1-1/2 inch	3

Note: The Global Edit tool found in the Edit menu can be used to enter the data for pipes P4-P11 all at once. The Guide at the top right of the window can be used to walk you through how to global edit if needed.

Junction Properties

1. J1 - J8 Spray Discharges
 1. Elevation = 3 meters
 2. Loss Model = Cd Spray (Discharge Coefficient)
 3. Geometry = Spray Nozzle
 4. Exit Properties = Pressure
 5. Exit Pressure = 0 barG (0 kPa (g))
 6. Cd (Discharge Coefficient) = 0.6
 7. Discharge Flow Area = 3.5 cm²
2. J9 Assigned Pressure
 1. Elevation = 0.3 meters
 2. Pressure = 14 barG (1400 kPa (g))
 3. Pressure Specification = Stagnation
3. J10 Branch
 1. Elevation = 3 meters
4. J11 Branch & J12 Dead End
 1. Elevation = 0.3 meters

Note that the supply pressure is what we are trying to solve for, but we must supply a guess to start. We will need to iterate on the pressure at junction J9 to determine what the minimum pressure should be.

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

Open Output Control by selecting Output Control from the toolbar or Tools menu. Click the Junctions button, then add Vol. Flow Rate Net At Junction to the shown junction output. This output will show the volumetric flow rate discharged from the spray nozzle. Click OK to save the changes and close Output Control.

The Output window contains all the data that was specified in the Output Control window. In the Junction output window at the bottom of the page there is a special tab for Spray Discharge, as shown in Figure 2. Select this tab and examine the flow rate out of each of these junctions. All flow rates are above 25 m³/hr so we must decrease the supply pressure and re-run the model. You should find that the supply pressure must be above 13.2 barG (1320 kPa-g) for the flow rate to meet the minimum flow rate at each spray nozzle.

This example may be solved directly using the [GSC Module](#).

Note: The parameter Volumetric Flow Rate Through Junction is for junctions with a single inlet and outlet, and will thus display N/A for the Spray Discharge nozzles that aren't at the end of the branch. The parameter Volumetric Flow Rate Junction Net is used instead to display the total volumetric flow rate that is exiting each spray nozzle.

⌵ All Junctions Assigned Pressure Branch Dead End Spray Discharge									
Spray Discharge	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m ³ /hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)	Vol. Flow Rate Jct Net (m ³ /hr)
1	Spray Discharge	6.666	1.013	6.814	1.013	25.77	7.148	39.12	25.77
2	Spray Discharge	N/A	1.013	7.046	1.013	N/A	N/A	39.12	26.28
3	Spray Discharge	N/A	1.013	7.968	1.013	N/A	N/A	39.12	28.21
4	Spray Discharge	N/A	1.013	10.138	1.013	N/A	N/A	39.12	32.32
5	Spray Discharge	N/A	1.013	10.138	1.013	N/A	N/A	39.12	32.32
6	Spray Discharge	N/A	1.013	7.968	1.013	N/A	N/A	39.12	28.21
7	Spray Discharge	N/A	1.013	7.046	1.013	N/A	N/A	39.12	26.28
8	Spray Discharge	6.666	1.013	6.814	1.013	25.77	7.148	39.12	25.77

Figure 2: Junction Output for Spray Discharge System with the supply pressure set to 14 barG (1400 kPa(g))

Figure 3 below displays the results for the final supply pressure of 13.2 barG (1,320 kPa(g)).

Spray Discharge System

Spray Discharge	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m ³ /hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)	Vol. Flow Rate Jct Net (m ³ /hr)
1	Spray Discharge	6.334	1.013	6.474	1.013	25.00	6.936	39.12	25.00
2	Spray Discharge	N/A	1.013	6.693	1.013	N/A	N/A	39.12	25.50
3	Spray Discharge	N/A	1.013	7.561	1.013	N/A	N/A	39.12	27.38
4	Spray Discharge	N/A	1.013	9.605	1.013	N/A	N/A	39.12	31.36
5	Spray Discharge	N/A	1.013	9.605	1.013	N/A	N/A	39.12	31.36
6	Spray Discharge	N/A	1.013	7.561	1.013	N/A	N/A	39.12	27.38
7	Spray Discharge	N/A	1.013	6.693	1.013	N/A	N/A	39.12	25.50
8	Spray Discharge	6.334	1.013	6.474	1.013	25.00	6.936	39.12	25.00

Figure 3: Junction Output for Spray Discharge System with the supply pressure set to 13.2 barG (1320 kPa (g))

Step 6. Use the Isometric Grid

AFT Fathom allows the user to place pipe or junction objects anywhere in the Workspace. Objects are placed on a 2D grid by default, as was the case with this example. At times it may be convenient to demonstrate the three-dimensional nature of a system. For example, if you are building a model based on isometric reference drawings. AFT Fathom includes an isometric grid for these cases. The isometric grid has three gridlines that are offset by 60 degrees, representing the x, y, and z axes. Figure 4 shows the spray discharge system built on an isometric grid. The completed model file drawn in isometric can be seen in the Examples folder and is named Spray Discharge Isometric.

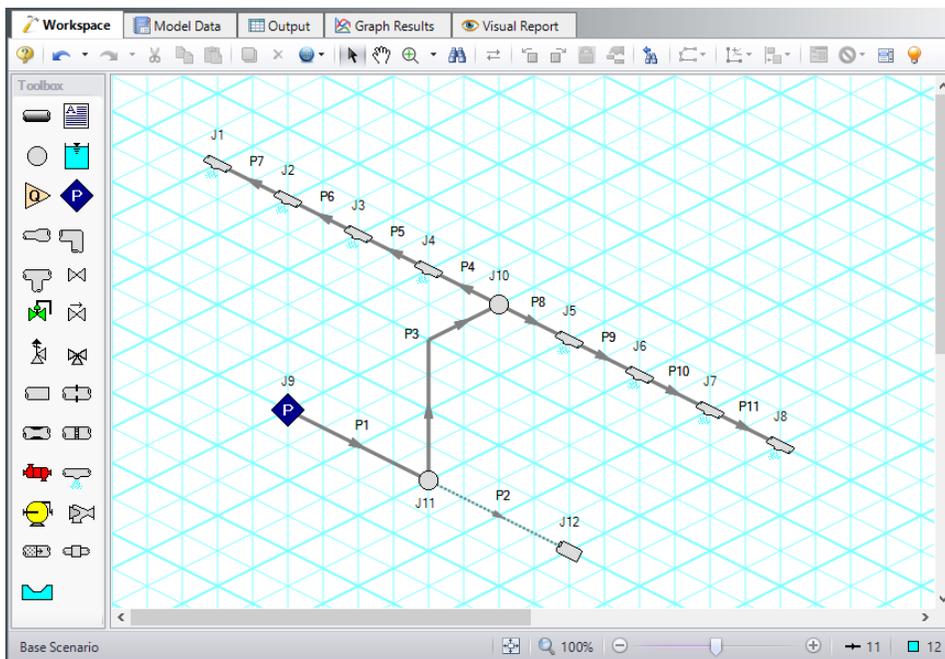


Figure 4: Spray discharge system with the isometric grid shown

You can enable the isometric grid in AFT Fathom by going to the Arrange menu. Under Pipe Drawing Mode, there are three options: 2D Freeform (default), 2D Orthogonal, and Isometric.

Creating the spray discharge system on an isometric grid will demonstrate how to use this feature:

1. Go to the File menu and select New.
2. Go to the Arrange menu again and under Pipe Drawing Mode, select Isometric
3. Place a Spray Discharge junction, J1, on the Workspace. You will notice that placing junctions onto the Workspace follows the usual rules, however, the visual appearance of the icon is more complex than in a 2D grid. Due to the increased number of axes, the preferred icon and rotation must be selected to obtain visual consistency.
4. Right click on J1 and select Customize Icon. Select the icon shown in Figure 5.

Note: If a junction in isometric mode has at least two pipes connected to it, the Auto-rotate Icon option can be selected instead of Customize Icon. Auto-rotate icon will automatically choose the icon orientation based on the connected pipes.

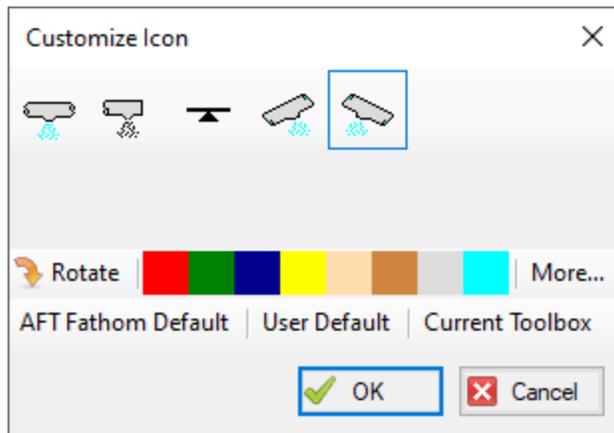


Figure 5: Right click on the junction to open the *Customize Icon* window and select the preferred icon and rotation

5. Copy J1 for the other spray discharges, J2-J8, to maintain the preferred icon and rotation.
6. Place the assigned pressure junction, J9, on the Workspace.
7. Place two branch junctions, J10 and J11, on the Workspace.
8. Place the dead end junction, J12, on the Workspace.
9. Right click on J12 and select Customize Icon. Notice that you may need to use the Rotate button to get the preferred icon and rotation, as shown in Figure 6.

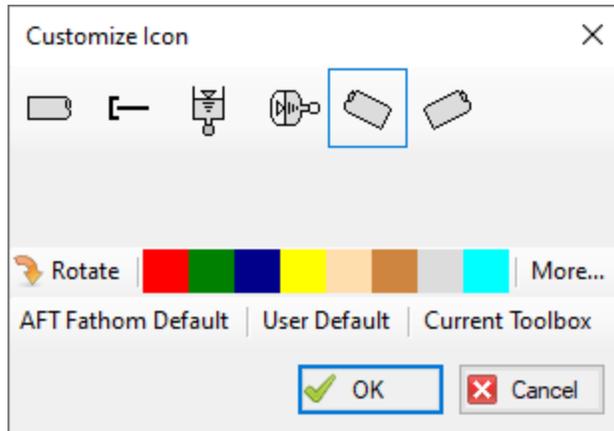


Figure 6: Use the Rotate button to obtain the preferred icon and rotation

10. Draw Pipes 1 and 2, as shown in Figure 4.
11. Draw Pipe 3 from J11 to J10. A red-dashed preview line will show how the pipe will be drawn on the isometric grid. As you are drawing a pipe, you can change the preview line by clicking any arrow key on your keyboard or scrolling the scroll wheel on your mouse.

Figure 7 shows Pipe 3 being drawn with the preview line.

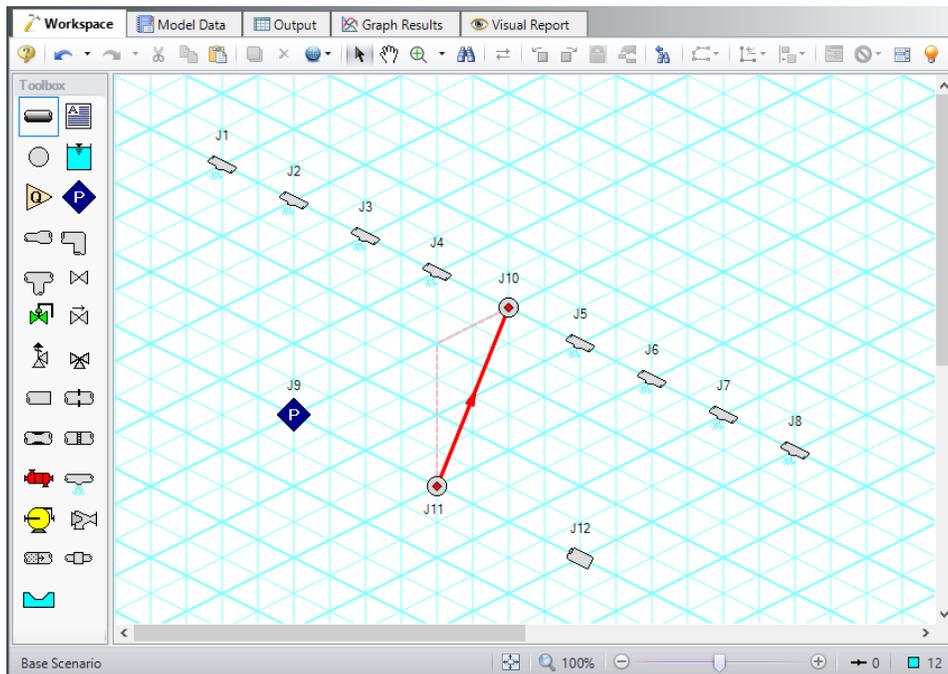


Figure 7: A preview line shows when drawing or adjusting pipes

Spray Discharge System

Note: You can hold the ALT key while adjusting a pipe by the endpoint to add an additional vertex. This can be used with the arrow key or mouse scroll wheel to change between different preview line options.

12. Draw Pipes 4-11, as shown in Figure 4.
13. The grid can be shown or turned off in the Arrange menu. Figure 8 shows the completed model with the grid not shown.

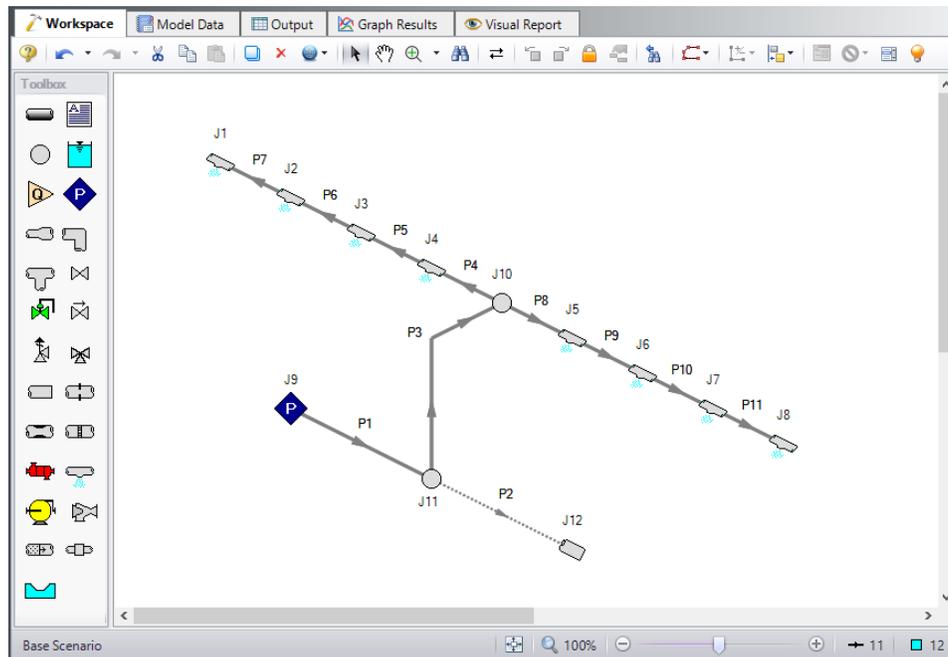


Figure 8: Isometric spray discharge system with the grid not shown

Pump Sizing and Selection with Flow Control Valves

Summary

This example will demonstrate how to size and select pumps in a heat exchanger system with flow control valves. The system will pump water through a heat exchanger system from a supply tank at one elevation and pressure to a receiving tank at a higher elevation and pressure. There will be two flow control valves operating in parallel. The control valves require a minimum pressure drop of 0.35 bar (35 kPa) in order to control the flow rate.

Topics Covered

- Using flow control valves
- Entering and changing pump data

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Pump Sizing and Selection with Flow Control Valves.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

- a. Fluid Library = AFT Standard
- b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
- c. Temperature = 21 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

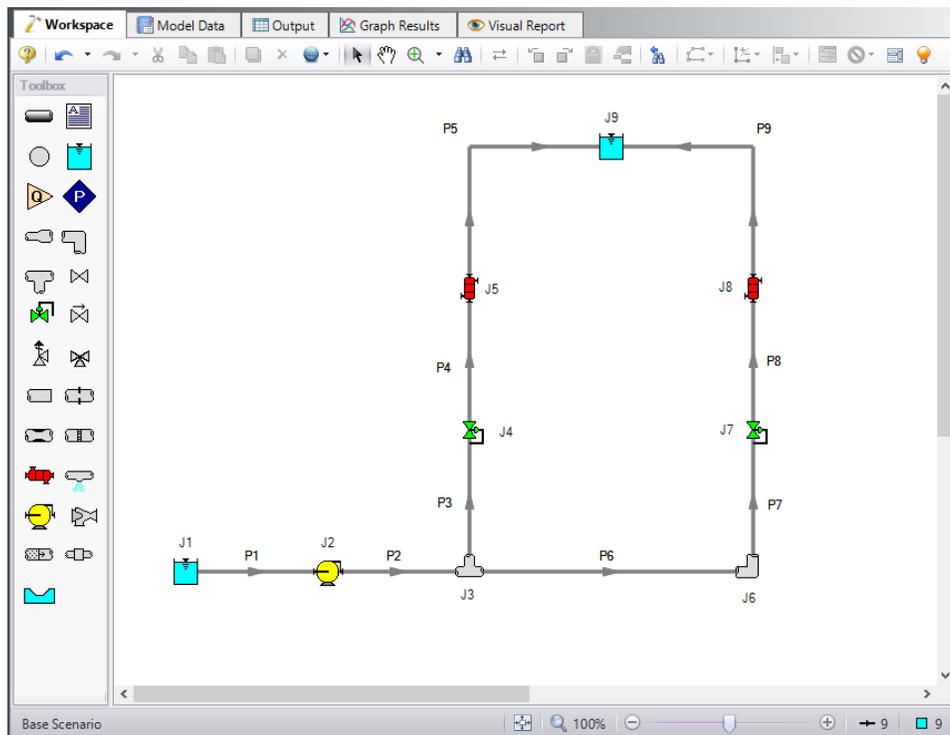


Figure 1: Workspace for Pump Selection with Flow Control Valves Example

The system is in place but now the properties of the objects must be entered.

Pipe Properties

1. Pipe Model tab
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 2 inch

- d. Type = STD (schedule 40)
- e. Friction Model Data Set = Standard
- f. Length =

Pipe	Length (meters)
1	6
2	15
3	20
4	6
5	30
6	6
7	20
8	6
9	30

Junction Properties

- 1. J1 Reservoir
 - a. Name = Supply Tank
 - b. Liquid Surface Elevation = 1.5 meters
 - c. Liquid Surface Pressure = 0.7 barG (70 kPa(g))
 - d. Pipe Depth = 1.5 meters
- 2. J2 Pump
 - a. Inlet Elevation = 0 meters
 - b. Pump Model = Centrifugal (Rotodynamic)
 - c. Analysis Type = Sizing
 - d. Parameter = Volumetric Flow Rate
 - e. Fixed Flow Rate = 50 m³/hr
- 3. J3 Tee
 - a. Elevation = 0 meters
 - b. Loss Model = Simple
- 4. J4 & J7 Control Valve
 - a. Inlet Elevation = 0 meters
 - b. Valve Type = Flow Control (FCV)
 - c. Control Setpoint = Volumetric Flow Rate
 - d. Flow Setpoint = 25 m³/hr
- 5. J5 & J8 Heat Exchanger

- a. Inlet Elevation = 0 meters
- b. Loss Model = Resistance Curve
- c. Enter Curve Data =

Volumetric	Pressure
m3/hr	bar
25	0.7

- d. Click Fill as Quadratic
 - e. Curve Fit Order = 2
 - f. Click Generate Curve Fit Now
- 1. J6 Bend
 - a. Inlet Elevation = 0 meters
 - b. Type = Standard Elbow (knee, threaded)
 - c. Angle = 90 Degrees
 - 2. J9 Reservoir
 - a. Name = Receiving Tank
 - b. Liquid Surface Elevation = 3 meters
 - c. Liquid Surface Pressure = 2 barG (200 kPa(g))
 - d. Pipe Depth = 3 meters

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Resolve Reference Pressure Needed

Click Run Model on the toolbar or from the Analysis menu.

The Reference Pressure Needed window will appear (see Figure 2). This window is displayed when a model is constructed that does not have a unique solution. A common occurrence of this is when a model contains one or more sections that are completely bounded by known flow rates.

In this case, because the pump has been designated as an assigned flow and the downstream control valves are flow control valves, there is no way to uniquely determine the pressure between the pump and the control valves without an additional reference pressure. The reference pressure is the pressure from which the other pressures in the system are derived. There can be more than one reference pressure, but there always has to be at least one.

For a more in-depth discussion of reference pressures, see the Fathom help file topic: [Role of Pressure Junctions - Detailed Discussion](#)

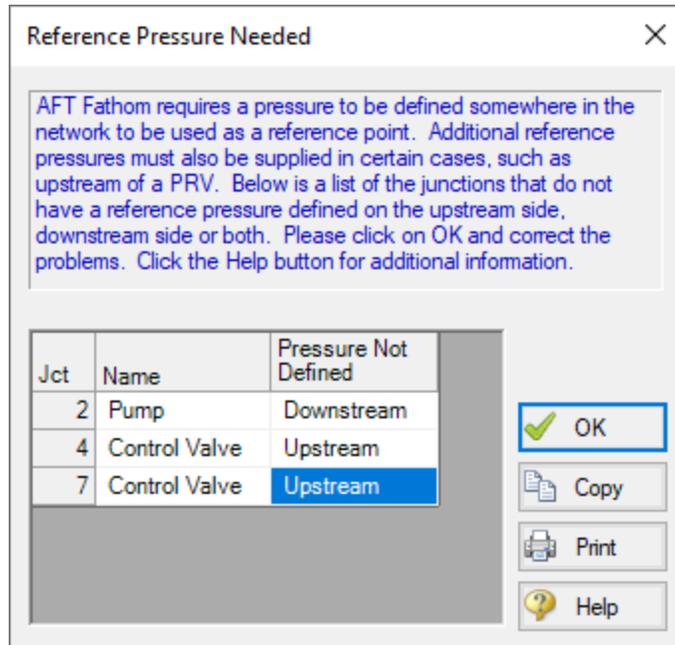


Figure 2: Reference Pressure Needed window

Open the J7 Control Valve Property window, and make the following changes.

1. Valve Type = Constant Pressure Drop (PDCV)
2. Control Setpoint = Pressure Loss
3. Pressure Drop = 0.35 bar (35 kPa)

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

We can look at the pump summary and see that the head at 50 m³/hr needs to be at least 55.35 meters (see Figure 3). The control valves both have enough pressure drop to control flow (see Figure 4).

Jct	Results Diagram	Name	Vol. Flow (m ³ /hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m ³ /hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
2	Show	Pump	50.00	13.87	5.421	55.35	100.0	N/A	7.524	N/A	N/A	13.96	N/A

Figure 3: Pump Summary window

Jct	Name	Valve Type	Vol. Flow (m3/hr)	Mass Flow (kg/sec)	dP Stag. (bar)	dH (meters)	P Static In (bar)	Cv	K	Valve State
4	Control Valve	FCV	25.00	6.936	0.5008	5.113	5.187	40.83	9.746	Open
7	Control Valve	PDCV	25.00	6.936	0.3500	3.574	5.036	48.84	6.812	Open

Figure 4: Valve Summary window

Step 7. Enter a Pump Curve

Now we need to specify a real pump. We will enter in some pump data and create a pump curve with the curve fitting routines in Fathom. Use the Scenario Manager on the Quick Access Panel to create a child of the base scenario with a name of Pump Curve. Then in the Pump Curve scenario, open the Pump Properties window and enter the following data:

1. J2 Pump
 - a. Pump Model = Centrifugal (Rotodynamic)
 - b. Analysis Type = Pump Curve
 - c. Enter Curve Data =

Volumetric	Head
m3/hr	meters
0	60
50	55
100	35

- c. Curve Fit Order = 2
2. J7 Control Valve
 1. Valve Type = Flow Control (FCV)
 2. Control Setpoint = Volumetric Flow Rate
 3. Flow Setpoint = 25 m3/hr

Re-run the model and verify that the flow rate and pressure drop across the control valves is still within the limits.

Freon Delivery System with Heat Transfer

Summary

This example will model a Freon delivery system to supply Freon at 24 deg. C. As the engineer you will evaluate two candidate refrigerants (Refrigerant 11 and 123) to determine the best choice. We will also calculate the temperature of the hot water supply (i.e., heat exchanger Secondary Fluid) needed for the heat exchangers.

Topics Covered

- Modeling heat transfer
- Entering a resistance curve for a Heat Exchanger

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Freon Delivery System with Heat Transfer.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

Freon Delivery System with Heat Transfer

- a. Fluid Library = NIST REFPROP
 - b. Fluid = R11 (after selecting, click Add to Model)
 - c. Fluid Phase = Liquid
 - d. Pressure = 10.5 barG
 - e. Temperature = -45 deg. C
3. Open the Heat Transfer/Variable Fluids panel
 - a. Select Heat Transfer With Energy Balance (Single Fluid)

This calculates the default fluid properties to use in the model to initialize the heat transfer calculations.

Note: When Heat Transfer is enabled the fluid properties in the Analysis Setup window are only used as an initial guess. The final fluid properties will be calculated by the Solver.

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

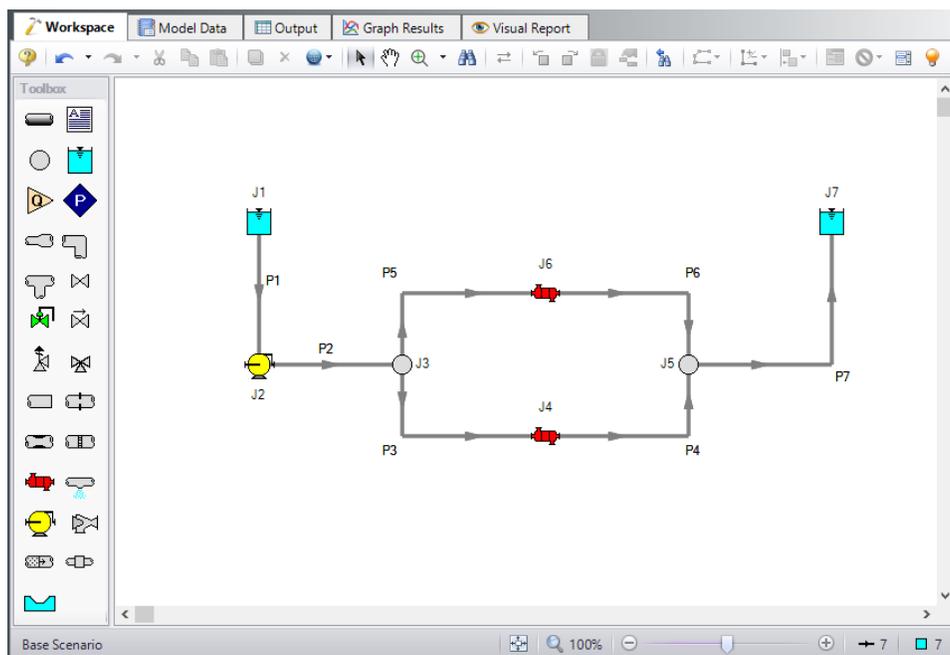


Figure 1: Model for Freon Delivery System

Now enter the properties for the pipes and the junctions.

Pipe Properties

1. Pipe Model tab

- a. Pipe Material = Steel - ANSI
- b. Pipe Geometry = Cylindrical Pipe
- c. Size = 3 inch
- d. Type = STD (schedule 40)
- e. Friction Model Data Set = Standard
- f. Lengths =

Pipe	Length (meters)
1 - 2	3
3 - 6	1.5
7	9

2. Fittings & Losses tab

- a. P3 - P6 only
 - i. Click Specify Fittings & Losses
 - ii. Bends tab
 - iii. Standard Threaded, General, 90 deg. (C) Quantity = 1
 - iv. Click OK

3. Heat Transfer tab (Figure 2)

- a. Heat Transfer Model = Convective Heat Transfer
- b. Resistance Geometry = Radial
- c. Number of Insulation Layers = 1 External
- d. Ambient Conditions
 - i. Temperature = 16 deg. C
 - ii. Fluid Velocity = 8 km/hr
 - iii. Fluid = Air
- e. Fluid Internal
 - i. Apply = Checked
 - ii. Conductivity/Convection Data Source = Correlation
 - iii. HT Area Ratio = 1
 - iv. Convection Coefficient = Gnielinski
- f. Pipe Wall
 - i. Apply = Checked
 - ii. Conductivity/Convection Data Source = Material Library
 - iii. Conductivity = From Library
 - iv. Thickness = 0.54864 cm
 - v. HT Area Ratio = 1
- g. Insulation #1 (external)
 - i. Apply = Checked
 - ii. Conductivity/Convection Data Source = User Specified

- iii. Conductivity = 0.7 W/m-K
 - iv. Thickness = 3 cm
 - v. HT Area Ratio = 1
- h. External
- i. Apply = Checked
 - ii. Conductivity/Convection Data Source = Correlation
 - iii. HT Area Ratio = 1
 - iv. Convection Coefficient = Forced (Churchill-Bernstein)

Pipe Properties

Number: 1 Upstream Junction: 1
 Name: Pipe Downstream Junction: 2
 Copy Data From Pipe... Copy Previous...

OK Cancel Jump... Help

Pipe Model Fittings & Losses **Heat Transfer** Design Alerts Optional Notes Status

Heat Transfer Model: Convective Heat Transfer
 Resistance Geometry: Radial
 Number of Insulation Layers: 1 External

Ambient Conditions
 Temperature: 16 deg. C
 Fluid Velocity: 8 km/hr
 Fluid: Air Water

Load Default

Resistance Type	Apply	Conductivity/Convection Data Source	Conductivity	Thickness	HT Area Ratio	Convection Coefficient
Units			W/m-K	cm		W/m2-K
Fluid Internal	<input checked="" type="checkbox"/>	Correlation			1	Gnielinski
Pipe Wall	<input checked="" type="checkbox"/>	Material Library	From Library	0.54864	1	
Insulation #1 (external)	<input checked="" type="checkbox"/>	User Specified	0.7	3	1	
External	<input checked="" type="checkbox"/>	Correlation			1	Forced (Churchill-Bernstein)

Figure 2: Heat Transfer tab defined in the Pipe Properties window

Junction Properties

1. J1 Reservoir
 - a. Name = Supply Reservoir
 - b. Liquid Surface Elevation = 3 meters
 - c. Liquid Temperature = -45 deg. C
 - d. Update Density with Temperature = Checked
 - e. Liquid Surface Pressure = 10.5 barG (1050 kPa(g))
 - f. Pipe Depth = 0 meters
2. J2 Pump
 - a. Inlet Elevation = 0.5 meters
 - b. Pump Model = Centrifugal (Rotodynamic)

- c. Analysis Type = Pump Curve
- d. Enter Curve Data =

Volumetric	Head
m3/hr	meters
0	7.5
45	7
115	4.5

- d. Curve Fit Order = 2
- 3. J3 & J5 Branches
 - a. Elevation = 0.5 meters
- 4. J4 & J6 Heat Exchangers
 - a. Inlet Elevation = 0.5 meters
 - b. Loss Model tab
 - i. Loss Model = Resistance Curve
 - ii. Enter Curve Data =

Volumetric	Pressure
m3/hr	bar
55	0.15

- iii. Click Fill As Quadratic
- iv. Curve Fit Order = 2
- v. Click Generate Curve Fit Now
- c. Thermal Data tab
 - i. Thermal Model = Counter-Flow
 - ii. Heat Transfer Area = 7.5 meters²
 - iii. Overall Heat Transfer Coefficient = 2800 W/m²-K
 - iv. Flow Rate = 15 kg/sec
 - v. Specific Heat = 4.2 kJ/kg-K
 - vi. Inlet Temperature = 95 deg. C

Note: This secondary fluid inlet temperature will be the initial guess.

- 5. J7 Reservoir
 - a. Name = Receiving Reservoir
 - b. Liquid Surface Elevation = 3 meters
 - c. Liquid Temperature = 24 deg. C
 - d. Update Density with Temperature = Checked

Freon Delivery System with Heat Transfer

- e. Liquid Surface Pressure = 10.5 barG (1050 kPa(g))
- f. Pipe Depth = 0 meters

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

When the Output is first shown, note that if the default Output Control parameters are being displayed, a warning will be shown stating the *HGL, EGL and head loss results may not be meaningful for variable density systems*. If all head output parameters are hidden using Output Control, this warning will no longer appear.

Open Output Control by selecting Output Control from the toolbar or Tools menu. Select the Pipes button to display the pipe output parameters, and remove the Head Loss (dH) from the list of shown output by highlighting the parameter and selecting Remove under the table. Next add the Temperature Inlet and Temperature Outlet to the shown output by double-clicking those parameters in the list on the left.

The Output window contains all the data that was specified in the output control window. The output from this model is shown in Figure 3. Now look at the outlet temperature of the last pipe (pipe 7). This is the temperature of the delivered Freon. The temperature is about 48.50 deg. C. This is significantly higher than the required temperature of 24 deg. C.

Pipe	Name	Vol. Flow Rate (m ³ /hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	T Inlet (deg. C)	T Outlet (deg. C)
1	Pipe	87.04	5.069	11.56	11.30	3.0000	0.5000	-0.25519	-0.25519	-0.4004	11.30	11.56	11.51	11.77	-45.00	-44.98
2	Pipe	87.03	5.069	12.48	12.33	0.5000	0.5000	0.14539	0.14539	0.0000	12.48	12.33	12.69	12.54	-44.98	-44.96
3	Pipe	43.52	2.534	12.49	12.44	0.5000	0.5000	0.04701	0.04701	0.0000	12.49	12.44	12.54	12.50	-44.96	-44.94
4	Pipe	50.01	2.913	12.34	12.29	0.5000	0.5000	0.05298	0.05298	0.0000	12.34	12.29	12.40	12.35	48.54	48.53
5	Pipe	43.52	2.534	12.49	12.44	0.5000	0.5000	0.04701	0.04701	0.0000	12.49	12.44	12.54	12.50	-44.96	-44.94
6	Pipe	50.01	2.913	12.34	12.29	0.5000	0.5000	0.05298	0.05298	0.0000	12.34	12.29	12.40	12.35	48.54	48.53
7	Pipe	99.97	5.822	12.11	11.27	0.5000	3.0000	0.83694	0.83694	0.3486	12.11	11.27	12.35	11.51	48.53	48.50

Figure 3: Pipe Output window for Freon Delivery System with water supply at 95 deg. C

Now iterate on the temperature of the secondary fluid in the heat exchangers to find the temperature required to produce an outlet temperature of 24.00 deg. C for pipe 7. The answer for this should be 57.8 deg. C. The output for this run is shown in Figure 4.

Freon Delivery System with Heat Transfer

Pipes		Heat Transfer														
Pipe	Name	Vol. Flow Rate (m3/hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	T Inlet (deg. C)	T Outlet (deg. C)
1	Pipe	87.23	5.080	11.56	11.30	3.0000	0.5000	-0.25456	-0.25456	-0.4004	11.30	11.56	11.51	11.77	-45.00	-44.98
2	Pipe	87.23	5.080	12.48	12.33	0.5000	0.5000	0.14602	0.14602	0.0000	12.48	12.33	12.69	12.54	-44.98	-44.96
3	Pipe	43.61	2.540	12.49	12.44	0.5000	0.5000	0.04722	0.04722	0.0000	12.49	12.44	12.54	12.49	-44.96	-44.94
4	Pipe	48.11	2.802	12.34	12.29	0.5000	0.5000	0.05119	0.05119	0.0000	12.34	12.29	12.40	12.35	24.01	24.01
5	Pipe	43.61	2.540	12.49	12.44	0.5000	0.5000	0.04722	0.04722	0.0000	12.49	12.44	12.54	12.49	-44.96	-44.94
6	Pipe	48.10	2.802	12.34	12.29	0.5000	0.5000	0.05120	0.05120	0.0000	12.34	12.29	12.40	12.35	24.01	24.01
7	Pipe	96.17	5.601	12.12	11.28	0.5000	3.0000	0.83562	0.83562	0.3632	12.12	11.28	12.35	11.51	24.01	24.00

Figure 4: Output window for Freon Delivery System with water supply at 57.8 deg. C

Now change the fluid on the Fluid panel to R123 and repeat the analysis. For R123, the temperature water supply to the heat exchangers necessary to meet the temperature requirement should be higher, around 66.5 deg. C. If water supply temperature is the main criteria, Refrigerant 11 would be the best choice.

Cooling System

Summary

The objective of this example is to determine how much flow will be delivered to the heat exchangers in the system. A secondary objective is to determine the best pump for the system.

Topics Covered

- Modeling a closed loop system
- Using a pressure reducing valve
- Generating a pump curve
- Generating a resistance curve for a heat exchanger
- Importing a pump curve to obtain pump data

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Cooling System.fth***
- ***Metric - Cooling System Pump A.txt***
- ***Metric - Cooling System Pump B.txt***
- ***Metric - Cooling System Pump C.txt***

Note: If you open the completed ***Metric - Cooling System.fth*** model file, you will notice a child scenario named **Initial Model for ANS Quick Start Chapter 3**. This child scenario is used as the starting point for a model in the ANS Quick Start Guide. This scenario can be disregarded when comparing your model to this completed model.

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

Cooling System

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = AFT Standard
 - b. Fluid = Dowtherm J (after selecting, click Add to Model)
 - c. Temperature = 65 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

To add the segment into pipe P22, use the Pipe Segments tool on the Arrange menu.

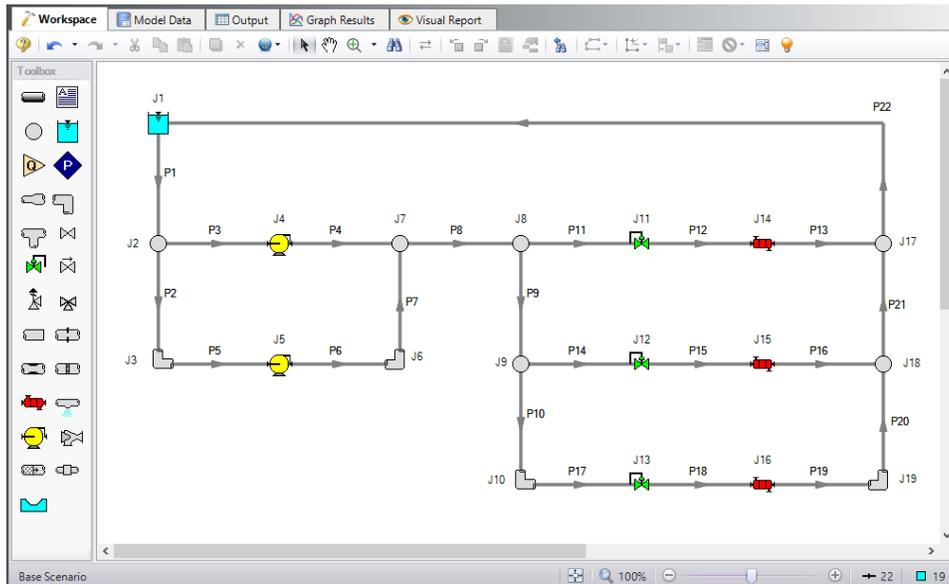


Figure 1: Workspace window for Cooling System Problem

Pipe Properties

1. Pipe Model tab
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = Use table below
 - d. Type = STD (schedule 40)
 - e. Friction Model Data Set = Standard
 - f. Lengths = Use table below

Pipe	Size	Length (meters)
1	6 inch	6
2, 7, 9-10, 20-21	4 inch	1
3-6, 12-13, 15-16, 18-19	4 inch	1.5
11, 14, 17	4 inch	3
8, 22	6 inch	60

Junction Properties

1. J1 Reservoir
 - a. Liquid Surface Elevation = 3 meters
 - b. Liquid Surface Pressure = 2.75 barG (275 kPa (g))
 - c. Pipe Depth = 3 meters
2. All Branches
 - a. Elevation = 0.5 meters
3. All Bends
 - a. Inlet Elevation = 0.5 meters
 - b. Type = Standard Elbow (knee, threaded)
 - c. Angle = 90 Degrees
4. J4 & J5 Pumps
 - a. Inlet Elevation = 0.5 meters
 - b. Pump Model = Centrifugal (Rotodynamic)
 - c. Analysis Type = Pump Curve
 - d. Enter Curve Data =

Volumetric	Head
m ³ /hr	meters
0	21.3

Volumetric	Head
m3/hr	meters
115	18.3
230	12.2

- e. Curve Fit Order = 2
- 5. All Control Valves
 - a. Inlet Elevation = 0.5 meters
 - b. Valve Type = Pressure Reducing (PRV)
 - c. Control Setpoint = Pressure, Static
 - d. Setpoint = 3.45 barG (345 kPa (g))
- 6. All Heat Exchangers
 - a. Inlet Elevation = 0.5 meters
 - b. Loss Model = Resistance Curve
 - c. Enter Curve Data =

Volumetric	Pressure
m3/hr	bar
115	0.7

- d. Click Fill As Quadratic
- e. Curve Fit Order = 2

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

The output is shown below in Figure 2. The pump summary shows that the head requirement for the pumps is 18.44 meters and 19.06 meters. The heat exchanger summary shows that the flow rates through the heat exchangers range from 66.60 m3/hr to 70.76 m3/hr.

Cooling System

General Warnings Design Alerts Pump Summary Valve Summary Heat Exchanger Summary Reservoir Summary													
Jct	Results Diagram	Name	Vol. Flow (m3/hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
4	Show	Pump	111.46	25.69	1.500	18.44	N/A	100.0	4.642	N/A	N/A	N/A	N/A
5	Show	Pump	94.80	21.85	1.550	19.05	N/A	100.0	4.079	N/A	N/A	N/A	N/A

All Junctions Bend Branch Control Valve Heat Exchanger Pump Reservoir								
Heat Exchanger	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
14	Heat Exchanger	4.457	4.191	4.481	4.215	70.85	16.33	11.1544
15	Heat Exchanger	4.457	4.207	4.480	4.229	68.79	15.85	11.1544
16	Heat Exchanger	4.458	4.222	4.479	4.243	66.69	15.37	11.1544

Figure 2: Output Summary Windows for Cooling System example

Step 6. Evaluate Other Pumps

The customer for this project wants an evaluation of three other pumps to determine which uses the least energy. The pump curve for each of these pumps is found in a text file.

For each of the pumps follow the following set of steps:

1. Create a child of the base scenario in the Scenario Manager on the Quick Access Panel for Pump A, Pump B, and Pump C.
2. Load scenario Pump A, open the Pump Properties window, click Enter Curve Data
3. Click Edit Table and select Import From File
4. Change the file format to Text (*.txt) then select the data file **Metric - Cooling System Pump A.txt** and click Open.
5. Using the Comma Separator, click OK.
6. With a Curve Fit Order of 2, click Generate Curve Fit Now, then click OK
7. Select the second pump and set all the properties equal to the first pump.

Run each scenario and take note of the overall pump power in the pump summary window. The data should match the following:

Pump	J4	J5
A	4.340 kW	3.428 kW
B	5.198 kW	4.112 kW
C	4.970 kW	3.296 kW

From this data, Pump A is the best choice, because it consumes the least power.

Housing Project

Summary

The objective of this example is to model a pipeline system to supply water to a new housing development. As the engineer you will evaluate the design to find the smallest pipe size that will still meet the water supply requirements.

No Fire Conditions:

Each house requires 2.5 m³/hr at a minimum of 4.0 barG (400 kPa(g)).

Fire Conditions:

Each house requires 0.5 m³/hr with no minimum pressure. One fire hydrant may be open at a time and requires 20 m³/hr at 6.2 barG (620 kPa(g)) static.

Topics Covered

- Using Scenario Manager

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Housing Project - Initial.fth***
- ***Metric - Housing Project.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **Metric - Housing Project- Initial.fth** model file listed above, which is located in the Examples folder in the AFT Fathom application folder. **Save the file to a different folder.**

The model should appear as shown in Figure 1.

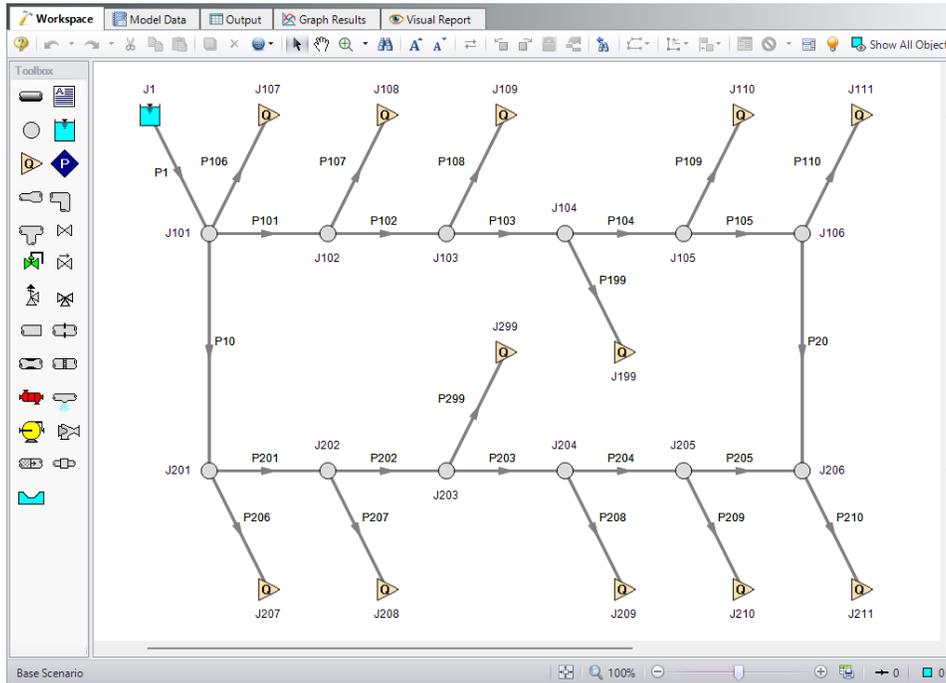


Figure 1: Initial model for Housing Project example

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = AFT Standard
 - b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
 - c. Temperature = 16 deg. C

Step 4. Define the Pipe and Junction Properties

The fire hydrants are both turned off to begin the problem. Select the Optional tab in the Assigned Flow Properties window and select Flow Off to close the hydrant. Do this for both hydrants.

Pipe Properties

All the mains are to be the same size. We will try 3, 3 ½, and 4 inches to see which one meets the requirements. Enter 3-inch pipe for the main and hydrant supply lines in the base case.

1. Pipe Model tab
 - a. Pipe Material = PVC - ASTM
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = Use table below
 - d. Type = schedule 40
 - e. Friction Model Data Set = Standard
 - f. Lengths = Use table below

Pipe	Size	Length (meters)
1	3 inch	610
10 & 20	3 inch	90
101-105 & 201-205	3 inch	60
199 & 299	3 inch	3
206	1 inch	45
108 & 207	1 inch	38
106, 110, 208, & 210	1 inch	30
107, 109, & 209	1 inch	22

Junction Properties

1. J1 Reservoir
 - a. Name = West Lake
 - b. Liquid Surface Elevation = 70 meters
 - c. Liquid Surface Pressure = 0 barG (0 kPa (g))
 - d. Pipe Depth = 0 meters
2. Branches
 - a. Elevation = Use table below

Housing Project

Branch	Elevation (meters)
203	-3
102-105 & 204	-1.5
101, 106, 202, & 205	0
201 & 206	1.5

3. Assigned Flows

- a. Elevation = Use table below
- b. Type = Outflow
- c. Flow Specification = Volumetric Flow Rate
- d. Flow Rate = Use table below

Assigned Flow	Name	Elevation (meters)	Flow Rate (m3/hr)	Special Conditions
107	Browns	3	2.5	None
108	Peters	6	2.5	None
109	Chambers	10	2.5	None
110	Fords	4.5	2.5	None
111	Kellys	6	2.5	None
199	North Hydrant	1.5	20	Flow Off
207	Norths	6	2.5	None
208	Stewarts	-4.5	2.5	None
209	Coles	9	2.5	None
210	Allens	3	2.5	None
211	Burns	1.5	2.5	None
299	South Hydrant	0	20	Flow Off

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

The flows will be varied based on the different scenarios examined. To do this, we need to create child scenarios. We will use the Scenario Manager to create the designs we need to evaluate.

Step 5. Manage the Scenarios

1. In the Scenario Manager, create a child of this scenario by right-clicking on the Base Scenario and selecting Create Child. Name this child scenario 3 inch Pipe. This will be the parent scenario for all the runs with 3 inch main pipes. Alternatively, you can select the Create Child icon  on the Scenario Manager on the Quick Access Panel to create this child scenario.
2. Select the 3 inch Pipe scenario and create another child, this one named Fire Flow. set the flow rate at each house to 0.5 m³/hr instead of 2.5 m³/hr. This will be the flow rate demand that needs to be met at the houses when either hydrant is open.
3. Next, create two children of this scenario named North Hydrant Fire and South Hydrant Fire. Load the North Hydrant Fire scenario from the Scenario Manager. Open the properties window of the North Hydrant and on the Optional tab, toggle the flow on by changing the special condition to None.
4. Now load the South Hydrant Fire scenario turn on the South Hydrant.
5. You now have all the scenarios that need to be evaluated for each pipe size but we still have two more pipe sizes that we need to investigate.
6. Right-click the 3 inch Pipe scenario and select Clone With Children.
7. Name this scenario 3.5 inch Pipe.
8. Repeat this process to create another set of scenarios named 4 inch Pipe. The Scenario Manager on the Quick Access Panel should look like Figure 2.

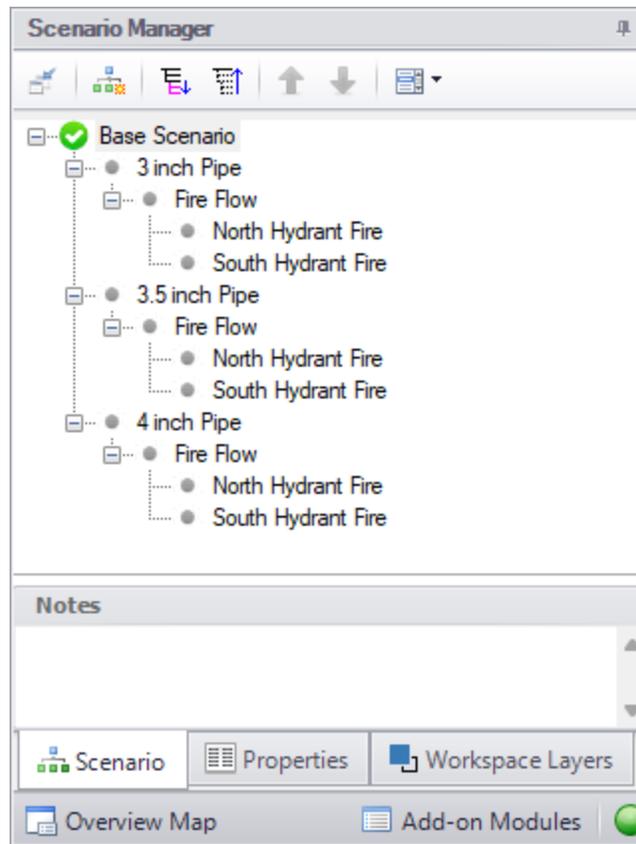


Figure 2: Scenario Manager for Water Supply to Housing Project

After creating the scenarios for the 3.5 inch pipe and 4 inch pipe, we need to open these scenarios and change the pipe sizes. This is accomplished most quickly with the Global Pipe Edit window.

1. Load the 3.5 inch Pipe scenario.
2. Select Global Edit then Global Pipe Edit from the toolbar or Edit menu.
3. Click the radio button for Pipe Template at the top, if not already selected, and click Select Pipe Data.
4. Change the Size to 3-1/2 inch, click OK.
5. In the Parameters to Change (Select in List) area, check the box next to Nominal Size 3-1/2 inch.
6. Since we only want to apply this to certain pipes, select Special above the Pipe List.
7. Change the Selection Type to Pipe Input Properties.
8. For Pipe Property, select Pipe Inner Diameter.
9. For Range, enter From 2.9 to 3.1 inches.
10. Click Select Pipes and note that 15 of 25 pipes were selected, these are all the pipes that are currently 3 inches. Click OK.
11. Click Apply Selections. A message should appear to inform you that changes were successively applied.
12. Click OK.

Repeat this process for the 4 inch Pipe scenario but changing the pipe size to 4 inch.

Step 6. Specify Output

Before running the model, load the Base Scenario to set the Output Control. Open the Output Control window by selecting Output Control from the toolbar or Tools menu. This window allows the user to specify the output from the Fathom simulation. The window starts with a default set of outputs. Often the default output is sufficient. You may also specify a title for the model. Click on the General tab and add a title of **Housing Project Example**. On the Display Parameters tab, select the Junctions button and change the static and stagnation pressure units to barG. Click OK to save and close the Output Control

Now load the 3 inch Pipe scenario.

Step 7. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 8. Examine the Output

The output from this model is shown in Figure 3. For the 3 inch Pipe scenario, with no hydrants open, the flow to each house is 2.5 m³/hr and the static pressure must be at least 4 barG (400 kPa(g)). The pressure at each house is above 4 barG (400 kPa(g)), so the system with 3 inch mains would be sufficient.

⌵ All Junctions Assigned Flow Branch Reservoir								
Assigned Flow	Name	P Static In (barG)	P Static Out (barG)	P Stag. In (barG)	P Stag. Out (barG)	Vol. Flow Rate Thru Jct (m ³ /hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
107	Browns	4.858	4.858	4.866	4.866	2.500	0.6942	0
108	Peters	4.582	4.582	4.590	4.590	2.500	0.6942	0
109	Chambers	4.055	4.055	4.063	4.063	2.500	0.6942	0
110	Fords	4.677	4.677	4.684	4.684	2.500	0.6942	0
111	Kellys	4.469	4.469	4.476	4.476	2.500	0.6942	0
X199	North Hydrant	5.144	5.144	5.144	5.144	0.000	0.0000	0
207	Norths	4.411	4.411	4.419	4.419	2.500	0.6942	0
208	Stewarts	5.468	5.468	5.476	5.476	2.500	0.6942	0
209	Coles	4.177	4.177	4.185	4.185	2.500	0.6942	0
210	Allens	4.816	4.816	4.824	4.824	2.500	0.6942	0
211	Burns	4.908	4.908	4.916	4.916	2.500	0.6942	0
X299	South Hydrant	5.285	5.285	5.285	5.285	0.000	0.0000	0

Figure 3: Output window for 3 inch mains with no fire

On the Scenario Manager, load the North Hydrant Fire scenario under the 3 inch Pipe scenario. Run the model, and look at the output. This output is shown in Figure 4. Although the hydrant receives 20 m³/hr, the supply pressure is less than the required 6.2 barG (620 kPa(g)).

Housing Project

⤴ All Junctions Assigned Flow Branch Reservoir								
Assigned Flow	Name	P Static In (barG)	P Static Out (barG)	P Stag. In (barG)	P Stag. Out (barG)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
107	Browns	5.060	5.060	5.060	5.060	0.5000	0.1388	0
108	Peters	4.708	4.708	4.708	4.708	0.5000	0.1388	0
109	Chambers	4.251	4.251	4.251	4.251	0.5000	0.1388	0
110	Fords	4.752	4.752	4.752	4.752	0.5000	0.1388	0
111	Kellys	4.614	4.614	4.614	4.614	0.5000	0.1388	0
199	North Hydrant	5.034	5.034	5.040	5.040	20.0000	5.5539	0
207	Norths	4.723	4.723	4.724	4.724	0.5000	0.1388	0
208	Stewarts	5.734	5.734	5.734	5.734	0.5000	0.1388	0
209	Coles	4.374	4.374	4.374	4.374	0.5000	0.1388	0
210	Allens	4.948	4.948	4.948	4.948	0.5000	0.1388	0
211	Burns	5.076	5.076	5.076	5.076	0.5000	0.1388	0
X299	South Hydrant	5.289	5.289	5.289	5.289	0.0000	0.0000	0

Figure 4: Output window for 3 inch mains with north hydrant fire

Because we know that 3 inch is sufficient for No Fire conditions, we only need to test the Fire conditions for each pipe size. The results of this testing are displayed in Table 1. After testing all the fire conditions, 4 inch pipe is the only size that will deliver 6.2 barG (620 kPa(g)) to the North Hydrant during fire conditions. Therefore, the housing development mains must be 4 inch pipe.

Table 1: Static pressure in barG at the open hydrant for each combination of pipe size and active hydrant

Fire Flow Location	3 inch	3.5 inch	4 inch
North Hydrant Fire	5.034	5.878	6.258
South Hydrant Fire	5.166	6.018	6.401

Beginner - Cost Calculation

Summary

This example demonstrates how to set up cost calculations in AFT Fathom.

Determine the cost of the heat exchanger system over a 10-year period including material, installation, and energy costs.

Topics Covered

- Estimating simple costs
- Connecting cost libraries
- Using Cost Settings
- Using the Cost Report

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Beginner - Cost Calculation.fth***
- ***Controlled Heat Exchanger Temperature.dat*** - engineering library
- ***Controlled Heat Exchanger Temperature - Junction Costs.cst*** - cost library for ***Controlled Heat Exchanger Temperature.dat***
- ***Controlled Heat Exchanger Temperature - Pipe Costs.cst*** - cost library for **Steel - ANSI**

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

If you have already worked the Controlled Heat Exchanger Temperature example for either the GSC or XTS modules, you can open the file you built for those examples, create a new scenario for this example, and then skip to Step 4.

Important: The XTS and GSC modules and cost calculations can be performed in the same run to simulate control functions over time, but we are not going to do this here. Before proceeding, make sure to disable GSC by selecting the radio button for Ignore under GSC in the Modules panel in Analysis Setup. Also, make sure to disable XTS by selecting Steady Only in the XTS section.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = AFT Standard
 - b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
 - c. Temperature = 21 deg. C
 - d. This calculates the default fluid properties to use in the model to initialize the heat transfer calculations.
3. Open the Heat Transfer/Variable Fluidspanel:
 - a. Heat Transfer = Heat Transfer With Energy Balance (Single Fluid)

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

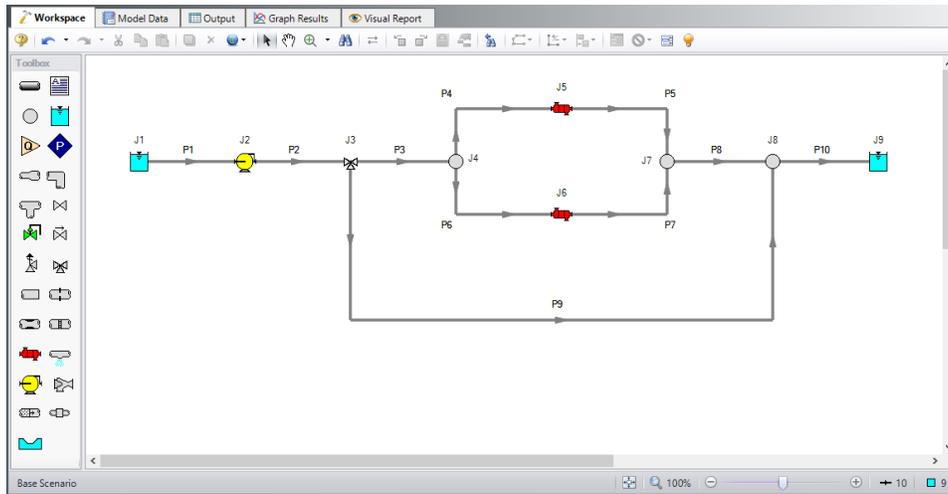


Figure 1: Workspace for Controlled Heat Exchanger Temperature example

Pipe Properties

1. All Pipes
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 8 inch
 - d. Type = STD (schedule 40)
 - e. Friction Model Data Set = Standard
 - f. Length =

Pipe	Length (meters)
1 & 3-8	6
2	1.5
9	60
10	30

Junction Properties

1. J1 & J9 Reservoirs
 - a. Liquid Surface Elevation = 6 meters
 - b. Liquid Temperature = 90 deg. C
 - c. Update Density with Temperature = Checked

- d. Liquid Surface Pressure = 0.3 barG (30 kPa (g))
- e. Pipe Depth = 0 meters

2. J2 Pump

- a. Inlet Elevation = 0 meters
- b. Pump Model = Centrifugal (Rotodynamic)
- c. Analysis Type = Pump Curve
- d. Enter Curve Data =

Volumetric	Head
m3/hr	meters
0	20
115	18
230	12

- e. Curve Fit Order = 2

3. J3 Three Way Valve

- a. Elevation = 0 meters
- b. Combined Flow = P2
- c. Split Flow Pipe #1 = P3
- d. Split Flow Pipe #2 = P9
- e. Loss Model = Cv
- f. Open Percentage Data =

Open Pct.	Cv Pipe #1	Cv Pipe #2
0	0	100
100	100	0

- g. Actual Percent Open = 75%

4. J4, J7, & J8 Branch Junctions

- a. Elevation = 0 meters

5. J5 & J6 Heat Exchangers

- a. Inlet Elevation = 0 meters
- b. Loss Model tab
 - i. Loss Model = K Factor
 - ii. K = 1400
- c. Thermal Data tab
 - i. Thermal Model = Counter-Flow
 - ii. Heat Transfer Area = 148 m²
 - iii. Overall Heat Transfer Coefficient = 280 W/m²-K
 - iv. Flow Rate = 90 kg/sec

- v. Specific Heat = 4.2 kJ/kg-K
- vi. Inlet Temperature = 4.5 deg. C

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Define the Cost Settings group

To perform cost calculations they will first need to be enabled. Do this by opening Analysis Setup, navigating to the Cost Settings panel, then selecting Calculate in the Cost Calculation section. If it is only desired to estimate energy costs, the Pump Energy Only option could be selected instead to simplify the input.

The Cost Settings panel is where the types of costs to be included in the cost calculations are specified. The Cost Settings panel is shown in Figure 2.

The Cost Definitions section allows you to specify the type of costs to be determined. You may select from engineering parameter costs, such as pipe weight, or actual monetary costs, including material and installation costs. For this example, select Monetary Costs, if not already selected. Include Material, Installation, and Operation/Energy costs by selecting them from the list of available monetary costs, as shown in Figure 2.

Energy cost data can be specified through an energy cost library, or by directly entering an energy cost on the Cost Settings panel. Using an energy cost library allows you to specify multiple energy costs, such as energy costs at both peak and off-peak rates. For this example, select Use This Energy Cost Information and enter a Cost of 0.12 U.S. Dollars Per kW-hr, as shown in Figure 2.

The system lifetime to be applied for the cost calculations is specified in the Cost Time Period section. You can also specify interest and inflation rates to be applied to the cost calculations as well. For this example, enter a system life of 10 years, as shown in Figure 2, then close the Cost Settings panel by clicking the OK button.

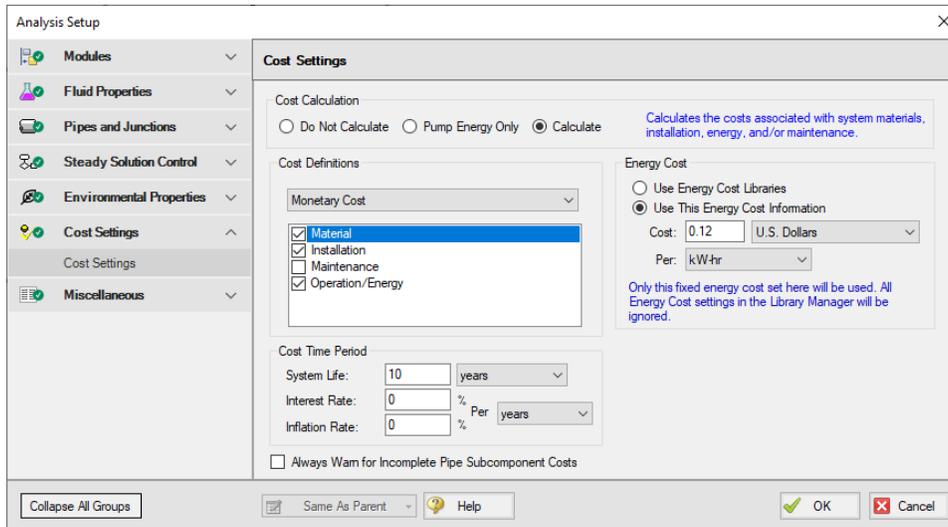


Figure 2: General cost calculation information is defined in the Cost Settings panel

Step 5. Engineering and Cost Libraries

The actual material and installation costs that would be used with cost calculations are contained in cost libraries. The cost libraries needed for this example already exist, and just need to be accessed.

The Library Manager (opened from the Library menu) shows all of the available and connected libraries. Libraries can either be engineering libraries or cost libraries. Cost libraries are always associated with an engineering library, and are thus displayed subordinate to an engineering library in the library lists.

Here we will summarize some key aspects of libraries:

- Cost information for a pipe is accessed from a cost library. Cost library items are based on corresponding items in an engineering library. (The engineering library also includes engineering information such as pipe diameters, hydraulic loss factors, etc.)
- To access a cost for a particular pipe or junction in a model, that pipe or junction must be based on items in an engineering library that is connected to the model file.
- There can be multiple cost libraries associated with and connected to an engineering library (such as regionally varying costs, or multiple foreign currencies). This makes it easier to manage costs of items.

Open the Library menu and select Library Manager. Click Add Existing Library, select the **Controlled Heat Exchanger Temperature.dat** engineering library from the AFT Fathom 13 Examples folder, and click Open. The engineering library titled **Controlled Heat Exchanger Temperature** should now appear in the list of available libraries.

Now add the cost libraries by clicking Add Existing Cost Library, and browsing to the **Controlled Heat Exchanger Temperature - Junction Costs.cst** and the **Controlled Heat Exchanger Temperature - Pipe Costs.cst** cost library files.

These libraries should connect automatically. If not, connect the newly added engineering and cost libraries by checking the box next to the name in the list of available libraries.

The Library Manager should appear as shown in Figure 3. The Library Manager window displays all of the available and connected libraries.

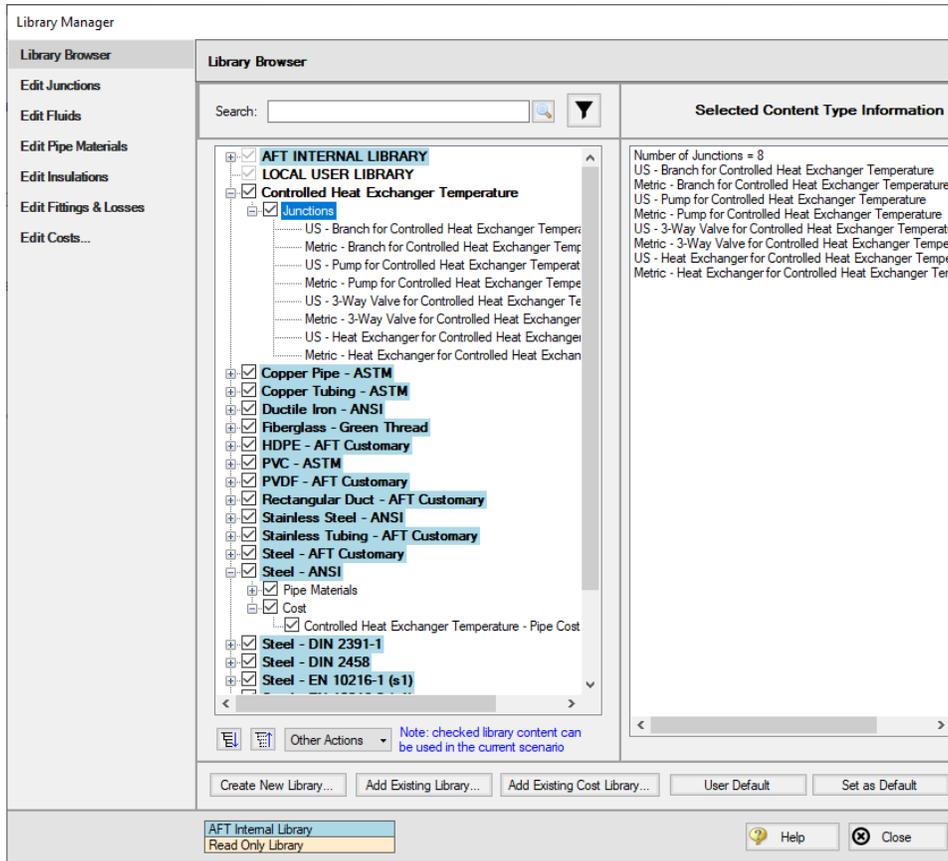


Figure 3: Library Manager with engineering and cost libraries added and connected

The engineering libraries associated with these two cost libraries are the **Steel - ANSI** pipe material library, and an external library called **Controlled Heat Exchanger Temperature**. For the cost data in the two cost libraries to be accessed by pipes and junctions in the model, the pipes and junctions must use these two engineering libraries.

The pipes for this example are already using data from the Steel - ANSI pipe material library, so no modifications need to be made for the pipes. All of the junctions except the reservoir junctions must be modified to use the junction data contained in the engineering libraries. Do this by opening the properties window for each junction, and selecting the junction name displayed in the Library list, as shown in Figure 4 for the Three-Way Valve Junction.

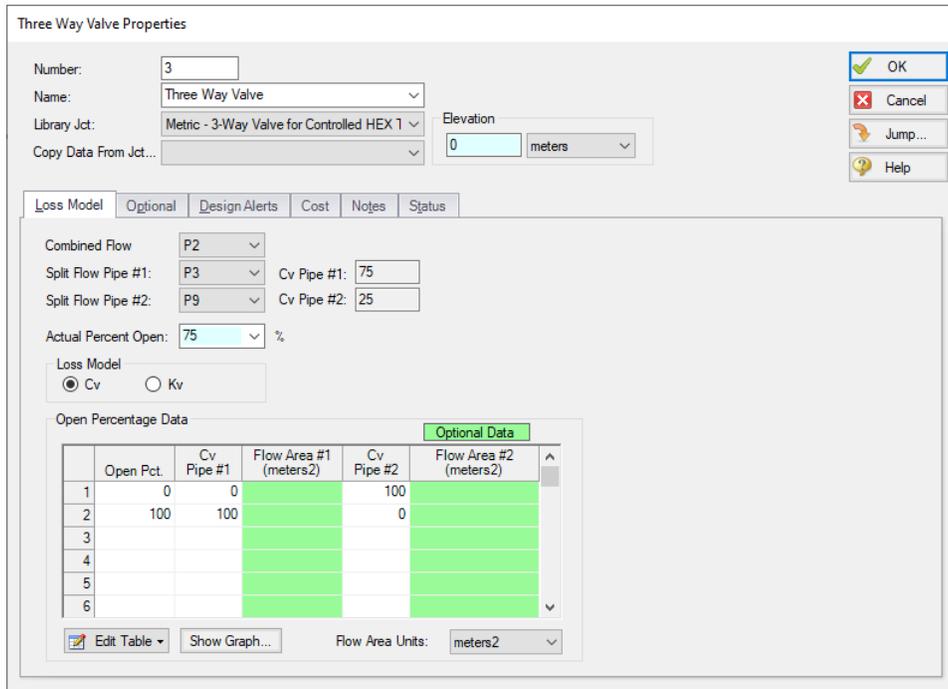


Figure 4: The junctions to be included in the cost analysis must be associated with an engineering library

Step 6. Including objects in the Cost Report

The final step before performing a cost analysis is to specify which pipes and junctions you want to be included in the final cost report. This is done by opening the properties window, navigating to the Cost tab, and choosing Include in Cost Report **for each pipe and junction to be included**, as shown for Pipe 1 in Figure 5.

The Global Edit feature may be used to update this information for all of the pipes and junctions.

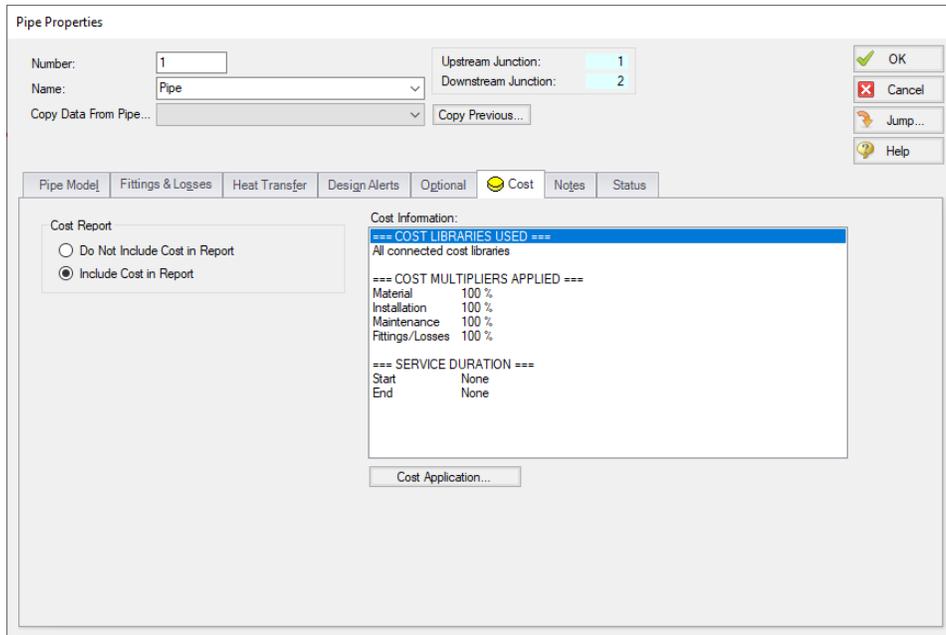


Figure 5: The Cost tab on the pipe and junction Properties windows is used to include the objects in the Cost Report

Step 7. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 8. Examine the Cost Report

The Cost Report is displayed in the General Results section of the Output Window. View the Cost report by selecting the Cost Report Tab. The content of the Cost Report can be controlled from the Output Control window. Figure 6 shows the Cost Report for this example with the Material, Installation, Energy, and Total costs displayed.

The Cost Report shows the total system cost, as well as the individual totals for the material, installation, and energy costs. In addition, the Cost Report displays the detailed cost for each pipe, junction, and fitting that was included in the report. The items are grouped together by type, and a subtotal for each category is listed.

Beginner - Cost Calculation

⌵ General Warnings Design Alerts Cost Report Pump Summary Valve Summary Heat Exchanger Summary Reservoir Summary									
Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Operation/Energy	Recurring Sub Total	TOTAL	
TOTAL OF ALL MODEL COSTS									99,001
Total of All Shown Costs			34,558	13,866	48,423	50,577	50,577	99,001	
Pipe Subtotal			22,338	10,950	33,287	0	0	33,287	
P1	Pipe	Pipe	1,004	492	1,496	0	0	1,496	
P2	Pipe	Pipe	251	123	374	0	0	374	
P3	Pipe	Pipe	1,004	492	1,496	0	0	1,496	
P4	Pipe	Pipe	1,004	492	1,496	0	0	1,496	
P5	Pipe	Pipe	1,004	492	1,496	0	0	1,496	
P6	Pipe	Pipe	1,004	492	1,496	0	0	1,496	
P7	Pipe	Pipe	1,004	492	1,496	0	0	1,496	
P8	Pipe	Pipe	1,004	492	1,496	0	0	1,496	
P9	Pipe	Pipe	10,039	4,921	14,961	0	0	14,961	
P10	Pipe	Pipe	5,020	2,461	7,480	0	0	7,480	
Branch Subtotal			1,620	666	2,286	0	0	2,286	
J4	Branch	Branch	540	222	762	0	0	762	
J7	Branch	Branch	540	222	762	0	0	762	
J8	Branch	Branch	540	222	762	0	0	762	
Heat Exchanger Subtotal			6,000	1,200	7,200	0	0	7,200	
J5	Heat Exchanger	Heat Exchanger	3,000	600	3,600	0	0	3,600	
J6	Heat Exchanger	Heat Exchanger	3,000	600	3,600	0	0	3,600	
Pump Subtotal			1,500	500	2,000	50,577	50,577	52,577	
J2	Pump	Pump	1,500	500	2,000	50,577	50,577	52,577	
Three Way Valve Subtotal			3,100	550	3,650	0	0	3,650	
J3	Three Way Valve	Three Way Valve	3,100	550	3,650	0	0	3,650	

Figure 6: The Cost Report in the General Output section shows the results of the cost analysis

Cost Analysis Summary

The cost analysis for this example shows the total system cost over 10 years for the heat exchanger system is \$99,001. Of that cost, \$34,558 was for Material costs, \$13,866 was for Installation costs, and \$50,577 was energy cost over the life of the system.

Non-Newtonian Phosphates Pumping

Summary

This example demonstrates how to perform Non-Settling Slurry calculations in AFT Fathom to size a pump as part of a system design process.

Topics Covered

- Entering rheological data
- Using homogeneous scale-up features
- Obtaining Power Law and Bingham Plastic constants

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Non-Newtonian Phosphates Pumping.fth***
- ***Metric - Non-Newtonian Phosphates Pumping Scale-up Data.txt***
- ***Metric - Non-Newtonian Phosphates Pumping Rheology Data.txt***

Problem Statement

The piping for a non-settling slurry system which moves phosphate is being designed. The system will pump 1,080 m³/hr of phosphate through a horizontal pipe 700 meters long.

Use non-settling slurry calculations to size the pump for 1,080 m³/hr and alternate potential flows of 720 and 1,440 m³/hr.

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Fluid panel
 - a. Fluid Library = User Specified Fluid
 - b. Name = Phosphate Slime
 - c. Density = 1130 kg/m³
 - d. Dynamic Viscosity = 1 centipoise (note that the viscosity entered here will *not* be used once the non-Newtonian parameters are entered).
3. Viscosity Model panel
 - a. Viscosity Model = Homogeneous Scale-up
 - b. Entered Scaled Data: The data provided by Wilson, et al, 2006 for 203 mm (8 inch) pipe is as follows:
 - i. Material = Steel - ANSI
 - ii. Size = 8 inch
 - iii. Type = STD (schedule 40)
 - iv. Raw Data table =

Table 1: Pressure drop data on phosphate slime from Wilson, et al, 2006 page 76 (203 mm/8 inch pipe)

Parameters	Velocity	Jm	Flow Regime
Units	meters/sec	m/m	
1	0.53	0.089	Laminar
2	1.52	0.100	Laminar
3	2.00	0.102	Laminar
4	2.59	0.105	Laminar
5	3.24	0.108	Laminar
6	3.81	0.110	Laminar
7	4.43	0.113	Laminar
8	5.12	0.119	Turbulent
9	5.64	0.130	Turbulent

1. In the table choose Velocity in meters/sec and Jm in m/m and use the Edit Table button to import the data from the ***Metric - Non-Newtonian Phosphates Pumping Scale-up Data.txt*** file. Alternatively, enter the data shown in Table 1 above, with the Flow Regime set to Laminar for the first 7

- entries, and Turbulent for rows 8 and 9.
2. Click Create Graph at the bottom.
3. To see how the data will scale to 305 mm, choose the **Cross-Plot Scale-Up Curves** option, with **Plot a Curve for Specific Pipe Size** selected, specify the following parameters: **Steel - ANSI, 12 inch, and STD**
4. Click Update Graph at the bottom (see Figure 2)
5. To see how Power Law and Bingham Plastic model would work for this data choose them in the **Cross-Plot Correlation for Test Pipe Diameter** area.
6. Select OK

Homogeneous Scale-up

Raw Data Graph

Test Pipe Material Data

Material: Steel - ANSI

Size: 8 inch

Type: STD (schedule 40)

Inner Diameter: 20.27174 cm

Parameter	Flow Velocity (meters/sec)	Head/Pressure Jm (m/m)		Flow Regime
	Units	Velocity	Jm	
1	meters/sec	0.53	0.089	Laminar
2		1.52	0.1	Laminar
3		2	0.102	Laminar
4		2.59	0.105	Laminar
5		3.24	0.108	Laminar
6		3.81	0.11	Laminar
7		4.43	0.113	Laminar
8		5.12	0.119	Turbulent
9		5.64	0.13	Turbulent
10				Turbulent
11				Turbulent
12				Turbulent
13				Turbulent

All Turbulent After First Turbulent

Figure 1: Non-Settling Slurry Scale-up test data entry

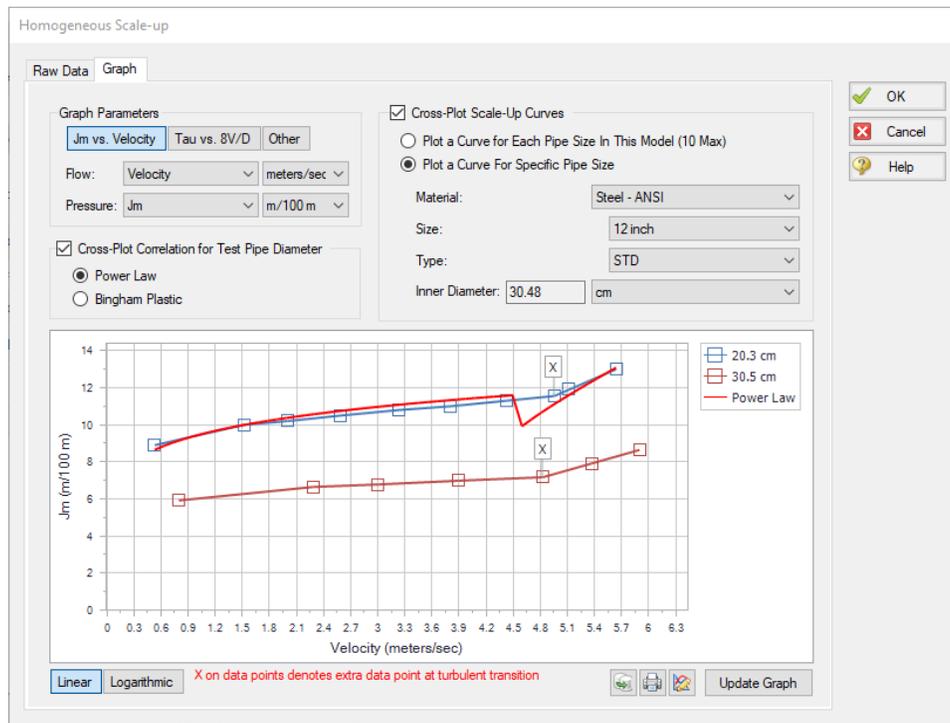


Figure 2: Graph of Non-Settling Slurry scale-up data

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

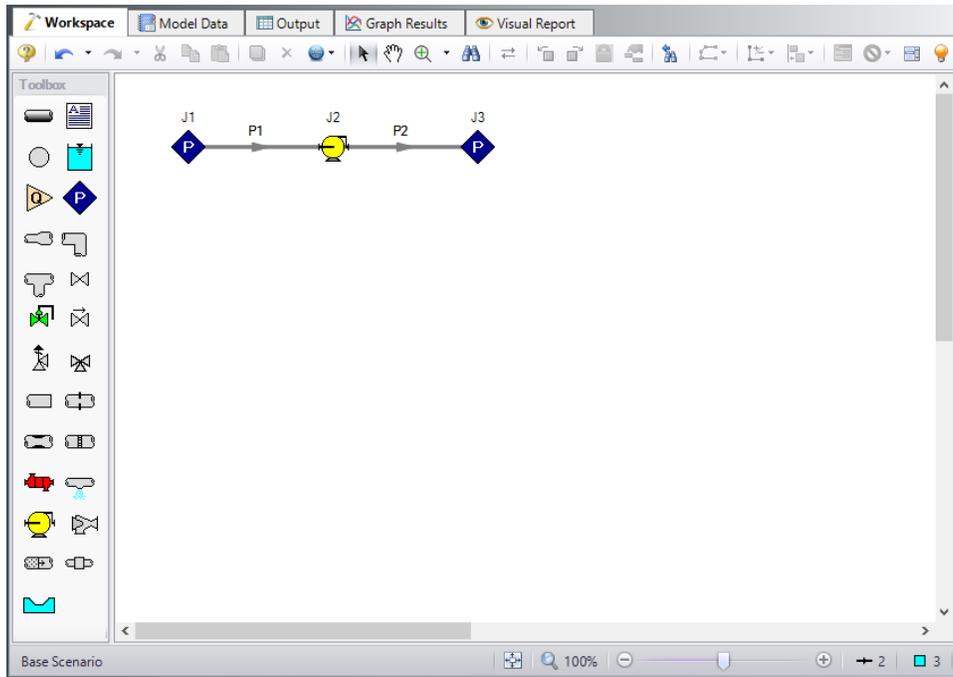


Figure 3: Layout of pipe system for Non-Newtonian Phosphates Pumping Example

Pipe Properties

1. Pipe Model tab
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 12 inch
 - d. Type = STD
 - e. Length =

Pipe	Length
1	1 cm
2	700 meters

Junction Properties

1. J1 & J3 Assigned Pressure
 - a. Elevation = 0 meters
 - b. Pressure = 0 barG (0 kPa(g))
 - c. Pressure Specification = Stagnation
2. J2 Pump

- a. Pump Model tab
 - i. Name = 1080 m³/hr
 - ii. Inlet Elevation = 0 meters
 - iii. Pump Model = Centrifugal (Rotodynamic)
 - iv. Analysis Type = Sizing
 - v. Parameter = Volumetric Flow Rate
 - vi. Fixed Flow Rate = 1080 m³/hr
 - vii. Nominal Efficiency = 75 Percent
- b. Optional tab
 - i. Additional Efficiency Data

Motor Power	Motor Eff. (%)
75	95

- ii. Power Units = kW

Step 4. Create Two Alternate Cases

The example in Wilson et al, 2006, also requires evaluation of alternate flow rates. To create the alternate cases do the following:

1. Go to the **Edit menu** and click **Select All**
2. From the **Edit menu** again, select **Duplicate Special**
3. Increment All Pipe and Junction Numbers By = **10**
4. Click **OK**
5. Move the duplicate below the first and click **Paste**
6. Repeat steps 2-5 on the pipes and junctions that were duplicated
7. For Pump J12, enter the following:
 - a. Name = 720 m³/hr
 - b. Fixed Flow Rate = 720 m³/hr
8. For the J22, enter the following:
 - a. Name = 1440 m³/hr
 - b. Fixed Flow Rate = 1440 m³/hr

To show the names of the pumps and their head rise, a new Workspace Layer will be created.

1. Select all pumps on the Workspace using a selection box (dragged from left to right) or holding CTRL while clicking each pump.
2. In the Quick Access Panel, open the Workspace Layers tab.
3. Create a new layer by selecting New Layer and then select the option From Workspace Selections in the drop-down menu. This creates a layer in which the selected objects and their labels are set to be visible.
4. Name the layer Pump Labels.

5. Edit the layer by selecting it and clicking the gear icon to bring up the Layer Settings window or by double clicking on the layer name.
6. Navigate to the Junction Parameters panel and expand the Commonly Used Junction Parameters list.
7. Double-click **Junction Name** to add it to the list on the right-hand side. This will add the Junction Name to the pump labels.
8. In the same panel, add **Head Rise (m)** to the label from the Pump Summary Parameters group.
9. Close the Layer Settings window.

Note: The Head Rise parameter will be grayed out in the right hand column since there is not yet output. After the model runs, this row will be white and each pump's head rise will be shown on the Workspace.

The workspace should resemble Figure 4.

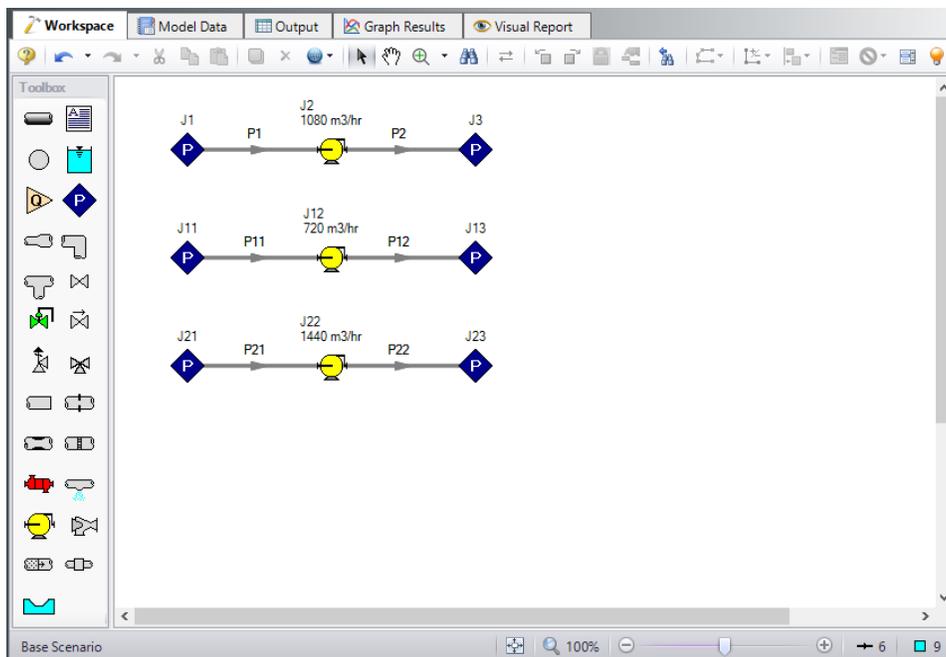


Figure 4: Layout of pipe system for Non-Newtonian Phosphates Pumping Example with all three cases

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

The Output window contains all the data that was specified in the Output Control window.

Figure 5 shows the Output window and includes the Pump Summary tab.

The screenshot shows the 'Output' window with three main data sections:

Pump Summary

Jct	Results Diagram	Name	Vol. Flow (m ³ /hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m ³ /hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
2	Show	1080 m ³ /hr	1,080.0	339.0	5.452	49.19	71.25	N/A	229.4	N/A	N/A	N/A	N/A
12	Show	720 m ³ /hr	720.0	226.0	5.224	47.14	71.25	N/A	146.5	N/A	N/A	N/A	N/A
22	Show	1440 m ³ /hr	1,440.0	452.0	6.261	56.50	71.25	N/A	351.3	N/A	N/A	N/A	N/A

Pipes

Pipe	Name	Vol. Flow Rate (m ³ /hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)
1	Pipe	1,080.0	4.112	0.9177	0.9177	0	0	0.00007788	0.00007788	0	0.0007028	0.9177	0.9177	1.013	1.013
2	Pipe	1,080.0	4.112	6.3692	0.9177	0	0	5.45145401	5.45145401	0	49.1941085	6.3692	0.9177	6.465	1.013
11	Pipe	720.0	2.741	0.9708	0.9707	0	0	0.00007463	0.00007463	0	0.0006735	0.9708	0.9707	1.013	1.013
12	Pipe	720.0	2.741	6.1950	0.9708	0	0	5.22422606	5.22422606	0	47.1435956	6.1950	0.9708	6.237	1.013
21	Pipe	1,440.0	5.482	0.8435	0.8434	0	0	0.00008945	0.00008945	0	0.0008072	0.8435	0.8434	1.013	1.013
22	Pipe	1,440.0	5.482	7.1046	0.8435	0	0	6.26116896	6.26116896	0	56.5010040	7.1046	0.8435	7.274	1.013

All Junctions

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m ³ /hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Assigned Pressure	0.9177	0.9177	1.013	1.013	1,080.0	339.0	0
2	1080 m ³ /hr	0.9177	6.3692	1.013	6.465	1,080.0	339.0	0
3	Assigned Pressure	0.9177	0.9177	1.013	1.013	1,080.0	339.0	0
11	Assigned Pressure	0.9708	0.9708	1.013	1.013	720.0	226.0	0
12	720 m ³ /hr	0.9707	6.1950	1.013	6.237	720.0	226.0	0
13	Assigned Pressure	0.9708	0.9708	1.013	1.013	720.0	226.0	0
21	Assigned Pressure	0.8435	0.8435	1.013	1.013	1,440.0	452.0	0
22	1440 m ³ /hr	0.8434	7.1046	1.013	7.274	1,440.0	452.0	0
23	Assigned Pressure	0.8435	0.8435	1.013	1.013	1,440.0	452.0	0

Figure 5: The Output window with default output data

The Output Control window can be used to customize the output and to add output parameters.

1. Open Output Control from the toolbar or Tools menu
2. In the Pipes section choose **Head Gradient** and add it to shown output. Then change its units to m/m. Head gradient is the same thing as *Jm*. See Figure 6.
3. Next choose **Pressure Gradient Stag. Total** and add it. Then change its units to bar/m.
4. Select the Pumps button that runs along the top of Output Control.
5. Choose **Pressure Rise Stagnation** and add it. Then change its units to m H₂O std.. This is the head rise in terms of water head. See Figure 7.
6. Click OK. The Output now appears as in Figure 8.

Table 3 summarizes the results from all three cases which agree well with those published by Wilson et al, 2006.

Non-Newtonian Phosphates Pumping

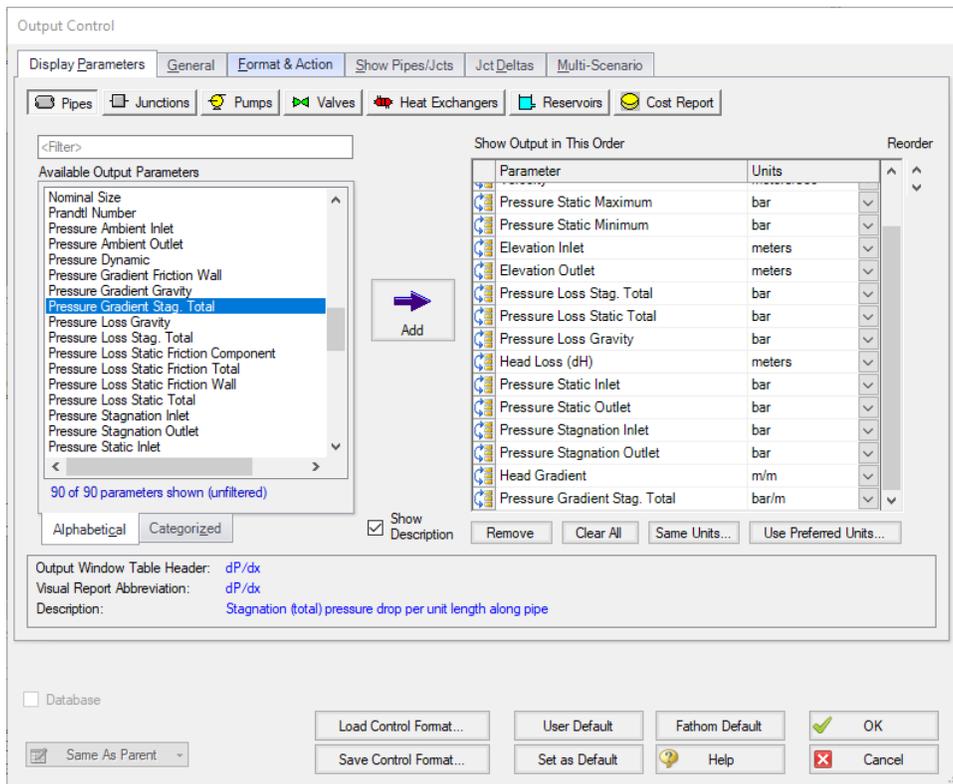


Figure 6: The Output Control window Pipes report

Non-Newtonian Phosphates Pumping

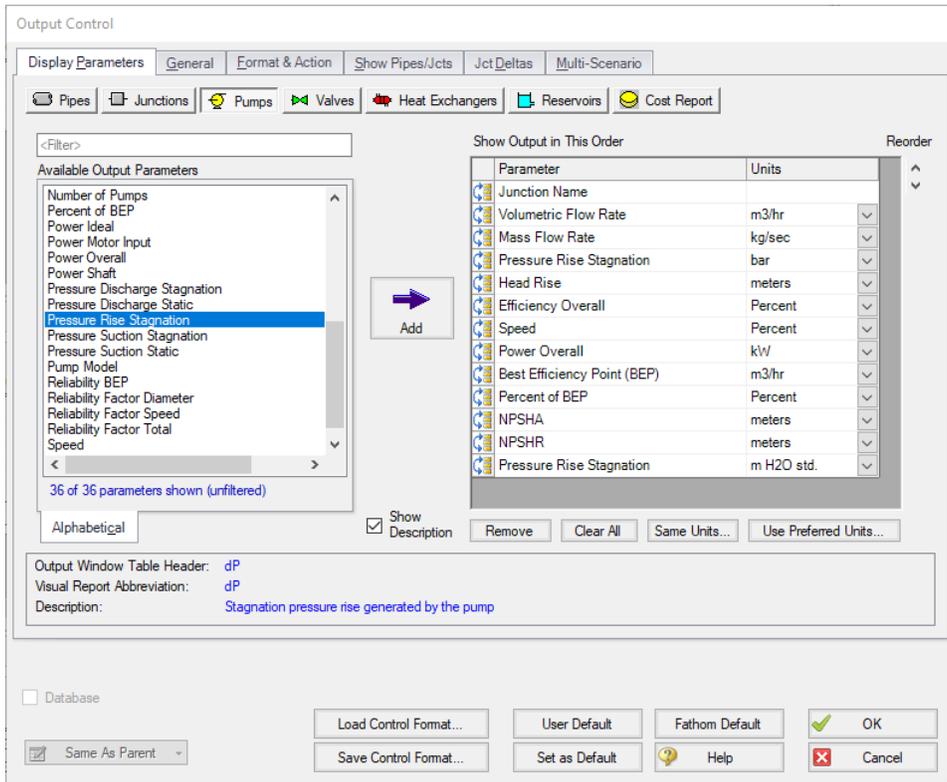


Figure 7: The Output Control window Pumps report

Workspace | Model Data | **Output** | Graph Results | Visual Report

General | Warnings | Design Alerts | Pump Summary

Jct	Results Diagram	Name	Vol. Flow Rate (m ³ /hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m ³ /hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)	dP (m H ₂ O std.)
2	Show	1080 m ³ /hr	1,080.0	339.0	5.452	49.19	71.25	N/A	229.4	N/A	N/A	N/A	N/A	55.59
12	Show	720 m ³ /hr	720.0	226.0	5.224	47.14	71.25	N/A	146.5	N/A	N/A	N/A	N/A	53.27
22	Show	1440 m ³ /hr	1,440.0	452.0	6.261	56.50	71.25	N/A	351.3	N/A	N/A	N/A	N/A	63.85

Pipes

Pipe	Name	Vol. Flow Rate (m ³ /hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	dH/dx (m/m)	dP/dx (bar/m)
1	Pipe	1,080.0	4.112	0.9177	0.9177	0	0	0.00007788	0.00007788	0	0.0007028	0.9177	0.9177	1.013	1.013	0.07028	7.788E-03
2	Pipe	1,080.0	4.112	6.3692	0.9177	0	0	5.45145401	5.45145401	0	49.1941085	6.3692	0.9177	6.465	1.013	0.07028	7.788E-03
11	Pipe	720.0	2.741	0.9708	0.9707	0	0	0.00007463	0.00007463	0	0.0006735	0.9708	0.9707	1.013	1.013	0.06735	7.463E-03
12	Pipe	720.0	2.741	6.1950	0.9708	0	0	5.22422606	5.22422606	0	47.1435956	6.1950	0.9708	6.237	1.013	0.06735	7.463E-03
21	Pipe	1,440.0	5.482	0.8435	0.8434	0	0	0.00008945	0.00008945	0	0.0008072	0.8435	0.8434	1.013	1.013	0.08072	8.945E-03
22	Pipe	1,440.0	5.482	7.1046	0.8435	0	0	6.26116896	6.26116896	0	56.5010040	7.1046	0.8435	7.274	1.013	0.08072	8.945E-03

All Junctions | Assigned Pressure | Pump

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m ³ /hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Assigned Pressure	0.9177	0.9177	1.013	1.013	1,080.0	339.0	0
2	1080 m ³ /hr	0.9177	6.3692	1.013	6.465	1,080.0	339.0	0
3	Assigned Pressure	0.9177	0.9177	1.013	1.013	1,080.0	339.0	0
11	Assigned Pressure	0.9708	0.9708	1.013	1.013	720.0	226.0	0
12	720 m ³ /hr	0.9707	6.1950	1.013	6.237	720.0	226.0	0
13	Assigned Pressure	0.9708	0.9708	1.013	1.013	720.0	226.0	0
21	Assigned Pressure	0.8435	0.8435	1.013	1.013	1,440.0	452.0	0
22	1440 m ³ /hr	0.8434	7.1046	1.013	7.274	1,440.0	452.0	0
23	Assigned Pressure	0.8435	0.8435	1.013	1.013	1,440.0	452.0	0

Base Scenario

Figure 8: The Output showing head gradient, pressure gradient and pump water head rise

Also of interest is the system curve. Navigate to the Graph Results window by clicking the Graph Results tab on the toolbar, or by pressing Control+G. On the Graph Control tab in the Quick Access Panel, select the Pump vs. System Curve tab if not already selected. CRV for Pump 2 should be selected, see Figure 9. Click Generate.

The system curve is shown in Figure 10.

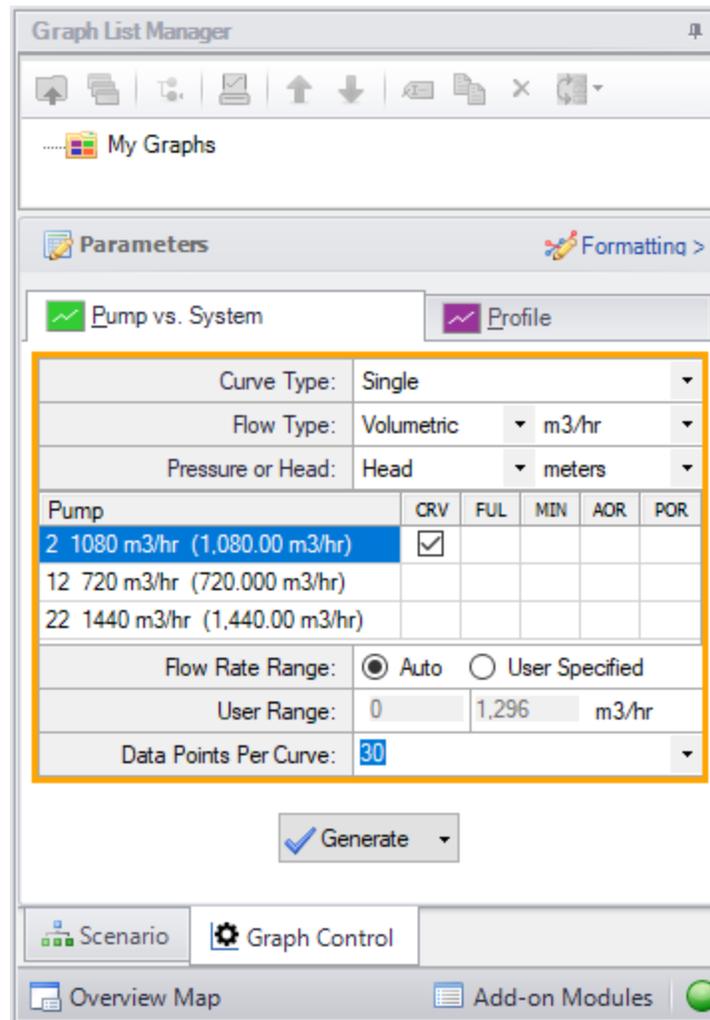


Figure 9: Data selection for graphing system curve

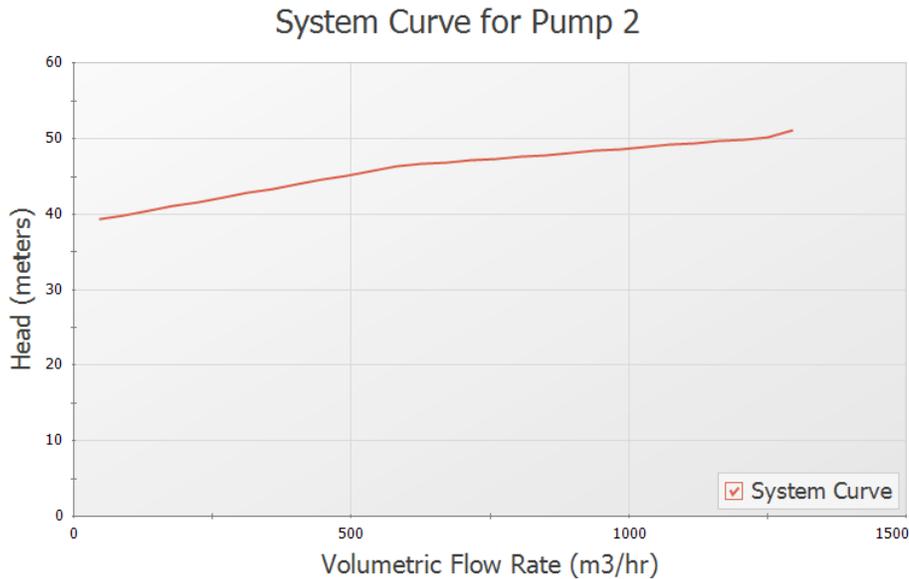


Figure 10: System curve in Graph Results window

Step 7. Compare Homogeneous Scale-Up to Power Law and Bingham Plastic Viscosity Models

Based on the fluid and flow regime and industry preference, a non-Newtonian model such as Power Law or Bingham Plastic may be preferred. How do those models compare to the Scale-up approach for the current example?

To perform the comparison, create two child scenarios. This can be done by opening the Scenario Manager on the Quick Access Panel. Create two child scenarios and call them Power Law and Bingham Plastic.

Here the Power Law scenario will be reviewed. The Bingham Plastic scenario will be very similar and hence is not shown. Results for all three scenarios will be compared later in this example.

1. Load the Power Law scenario
2. Open Analysis Setup
3. Open the Viscosity Model panel in the Fluid Properties group
4. Click the Enter Scaled Data button
5. Click the Graph tab
6. Select the Cross-Plot Correlation for Test Pipe Diameter option
7. Make sure the Power Law option is selected
8. If not, select it and click Update Graph

Figure 11 shows that the Power Law correlation matches the data well. The Bingham Plastic correlation diverges from test data at higher flow rates.

Non-Newtonian Phosphates Pumping

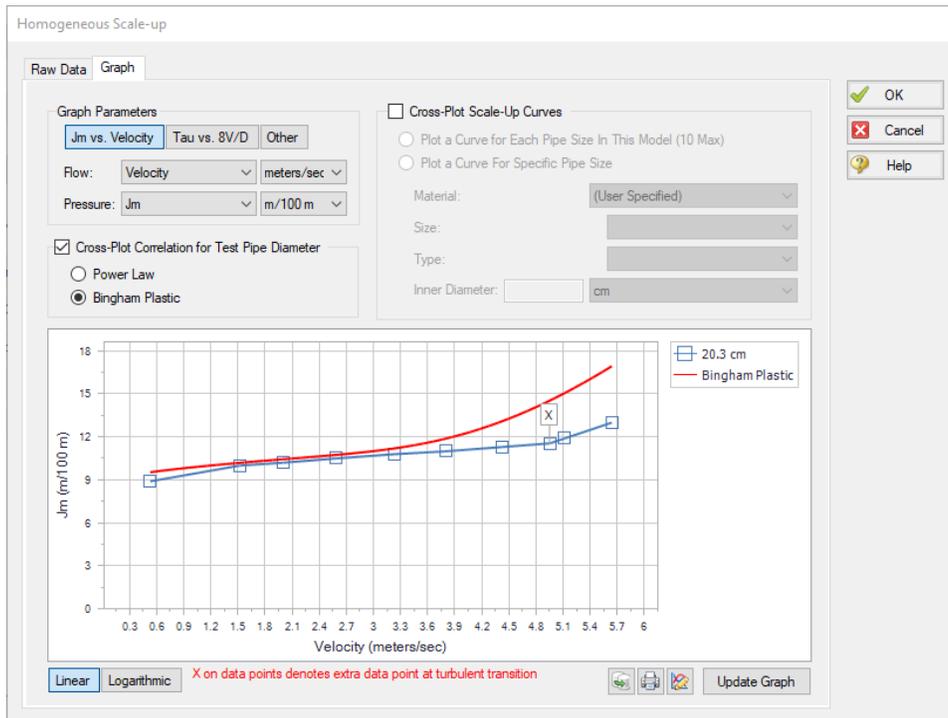
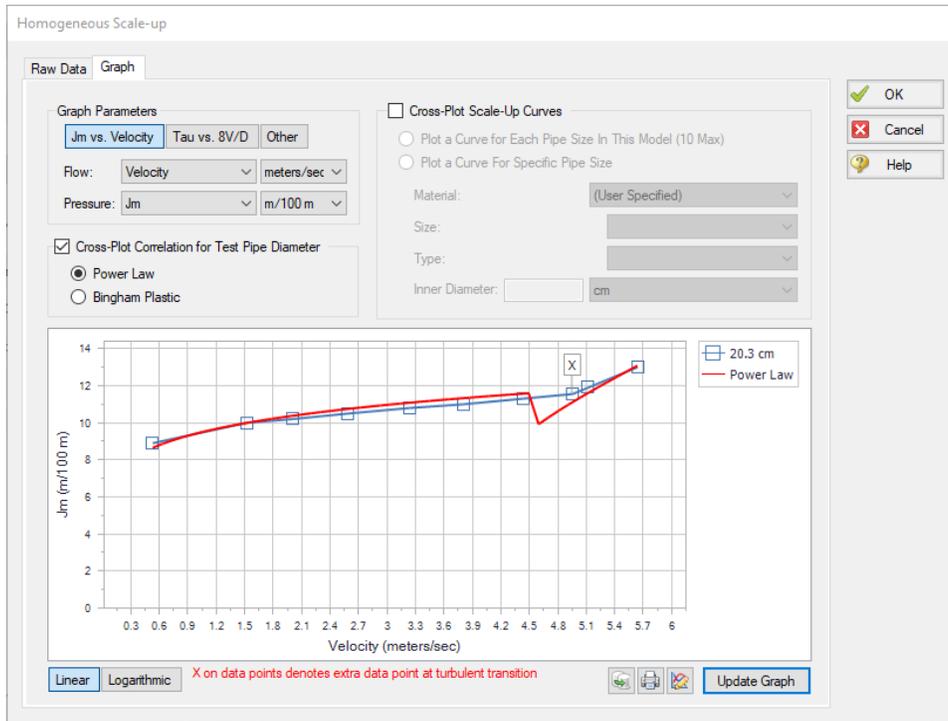


Figure 11: Non-Settling Slurry Scale-up data and Power Law (top) and Bingham Plastic (bottom) correlations

Non-Newtonian Phosphates Pumping

1. Click Cancel
2. Select Power Law from the drop down list of Viscosity Models
3. Select Calculate from Rheological Data
4. Click Calculate Constants
5. Make sure the Raw Data Type selection is Tube Flow Rheometer Data (8V/D)
6. Enter the shear rate and shear stress data from Table 2, below. This data is equivalent to the data from Table 1, entered previously. Alternatively, use the **Metric - Non-Newtonian Phosphates Pumping Rheological Data.txt** file to import the shear rate and shear stress data into Fathom.
7. Make sure the Shear Stress units are set to Pascals
8. Click Generate Curve Fit Now
9. Click OK

The data should be $K = 28.28 \text{ kg/sec-m}$ and $n = 0.1364$. See Figure 12. For the Bingham Plastic scenario the constants should be Plastic Viscosity= 0.03673 kg/sec-m and Yield Stress= 0.0004903 bar (0.04903 kPa).

Table 2: Pressure drop data on phosphate slime from Wilson, et al, 2006 page 76 (203 mm/8 inch pipe) converted into shear rate (8V/D) and shear stress (Tau)

Run	V	Jm	Shear Rate	Shear Stress
	meters/sec	m/m	1/seconds	Pascals
1	0.53	0.089	20.89	50.1
2	1.52	0.100	59.90	56.3
3	2.00	0.102	78.82	57.4
4	2.59	0.105	102.1	59.1
5	3.24	0.108	127.7	60.8
6	3.81	0.110	150.2	61.9
7	4.43	0.113	174.6	63.6
8	5.12	0.119	201.8	66.9
9	5.64	0.130	222.3	73.1

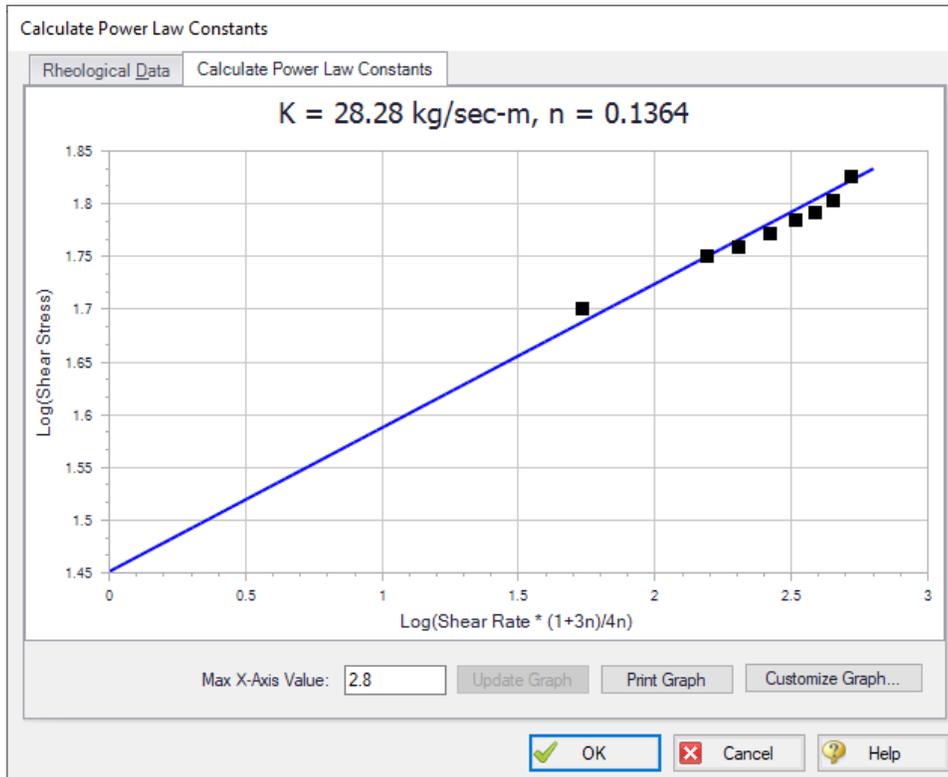


Figure 12: Power Law data curve fit

Each scenario can be loaded and run. Results are summarized in Table 3. It is clear that the Bingham Plastic model over predicts the required pump head and power at higher flow rates but is comparable at the lowest flow rate of 720 m3/hr. This is consistent with the lower part of Figure 11 where the Bingham Plastic correlation offers a poor match to the raw data as the flow rate increases.

Non-Newtonian Phosphates Pumping

Table 3: Comparison of results between Homogeneous Scale-up, Power Law, and Bingham Plastic models

Q	m3/hr	720 m3/hr			1080 m3/hr			1440 m3/hr		
		Scale-Up	Power Law	Bingham Plastic	Scale-Up	Power Law	Bingham Plastic	Scale-Up	Power Law	Bingham Plastic
V	meters/sec	2.741	2.741	2.741	4.112	4.112	4.112	5.482	5.482	5.482
Jm	m/m	0.0673-5	0.0683-0	0.06970	0.0702-8	0.0721-9	0.08067	0.0807-2	0.0821-6	0.10510
dP/dx	bar/m	7.463-E-03	7.569-E-03	0.0077-24	7.788-E-03	7.999-E-03	0.0089-39	8.945-E-03	9.105-E-03	0.0116-47
dP	bar	5.224	5.298	5.407	5.452	5.600	6.257	6.261	6.374	8.153
dH Water	meters	53.27	54.03	55.14	55.59	57.10	63.81	63.85	64.99	83.14
dH Fluid	meters	47.14	47.81	48.79	49.19	50.53	56.47	56.50	57.52	73.57
Power	kW	146.5	148.6	151.7	229.4	235.6	263.3	351.3	357.6	457.4

Analysis Summary

A non-settling phosphate slurry pipeline was modeled and the pump sized for a range of flow rates. AFT Fathom non-settling slurry calculations allow evaluation of the non-Newtonian viscous behavior of the slurry.

NFPA Firewater System

Summary

This problem walks through building a hydraulic model of a fire water sprinkler system using Fathom's NFPA features. This example is based off of figure B.2 (a) and associated calculations from NFPA 15 standard 8.5.3.1.

In this example, a firewater spray nozzle system is hydraulically modeled. The system consists of 28 spray nozzles and is supplied by a fixed pressure source. As the design engineer, it is your responsibility to ensure that all spray nozzles receive at least 152 L/min, as well as the minimum supply pressure required to achieve these flows. To ensure that the proposed design is adequate, you will model the system and use the hydraulically most remote nozzle to ensure that all spray nozzles are satisfied.

Topics Covered

- Modeling a fire water spray system
- Making groups
- Using NFPA features
- Generating an NFPA report
- Using Excel Change Data

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - NFPA Firewater System - Initial.fth***
- ***Metric - NFPA Firewater System - Final.fth***
- ***Metric - NFPA AFT Transfer Spreadsheet.xlsx***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the Model

Open the **Metric - NFPA Firewater System - Initial.fth** model file listed above, which is located in the Examples folder in the AFT Fathom application folder. **Save the file to a different folder.**

The model should appear as shown in Figure 1.

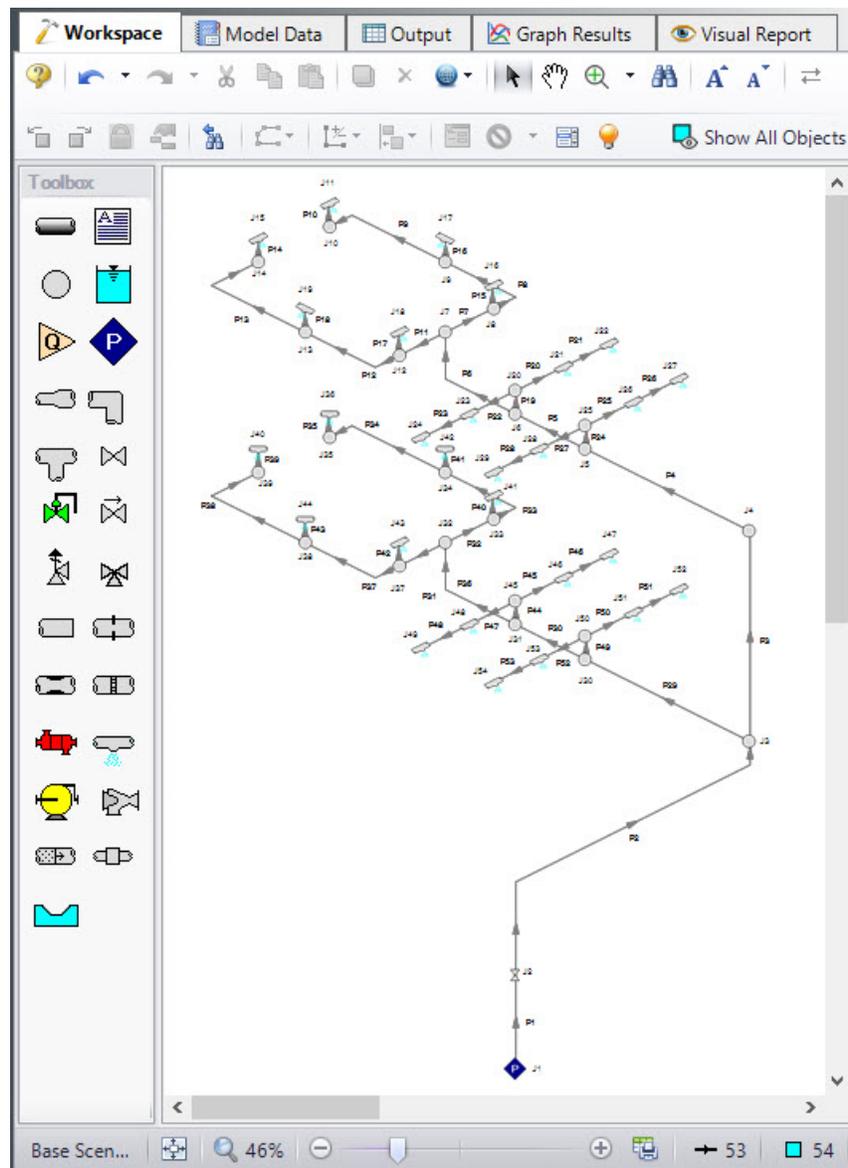


Figure 1: Initial model for NFPA Firewater System example

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = AFT Standard
 - b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
 - c. Temperature = 21 deg. C

Step 4. Define the Miscellaneous group

While still in the analysis setup, expand the Miscellaneous drop down and open the NFPA panel. Check the box next to Enable NFPA. This will enable various features throughout the software such as NFPA internal fittings & losses for pipes, equivalent length loss model for junctions, NFPA report, etc. Keep the Initial Values the same. A group for the NFPA report will be selected later.

Step 5. Define the Pipe and Junction Properties

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click the "Ok" button to save changes to the analysis setup and return to the workspace. At this point a user would need to draw their own network using the tools in the toolbox, but to save some time the piping layout for this model has already been created.

With the system in place, we now need to enter the properties of the objects. Because this model is large, all input has been added to a specially-configured [Excel Change data](#) spreadsheet that is included in the Examples folder and will be used to change the input to match the values in this example. Before importing this spreadsheet, it is necessary to define values in each pipe and junction because the Excel Change data spreadsheet is only capable of changing input data, not entering new data.

To quickly enter input data before using the Excel Change data spreadsheet, we will use the Global Pipe and Junction Editing features. If you are going to use Excel Change data to change the input you already have in your model, the values that you initially input in the model act simply as place holders and the value itself is not important. It is crucial, though, that you ensure that the units for each value that will be changed are specified correctly, because the Excel Change data spreadsheet will not change these units (it will only change the value itself).

Pipe Properties

To globally edit the pipes, select Global Pipe Edit from the Edit menu. Click All to select all pipes. Next, click Select Pipe Data and enter the following inputs in the pipe properties window.

1. Pipe Material = Steel - ANSI
2. Pipe Geometry = Cylindrical Pipe
3. Size = 1 inch
4. Type = STD (schedule 40)
5. Friction Model = User Specified
6. Data Set = Hazen-Williams Factor
7. Value = 120
8. Length = 3 meters

In the Parameters to Change section, click All to apply all changes to the selected pipes, then click Apply Selections at the bottom. Then click OK.

Next, fittings a losses will need to be added to various pipes throughout the system. On the fittings & losses tab of the pipe properties window move the slider from "Use K Factor" to "Use Equivalent Lengths" then specify the Fittings for each pipe according to the table below.

Note: Pipes that have Fittings & Losses entered in them will now have a red & displayed next to them. The symbol and display settings can be customized in User Options.

Table 1: Fittings and Losses for pipe input in NFPA model

Pipes	Fitting Type	Quantity
4, 6, 8, 10, 12, 14, 31, 33, 35, 37, 39	Elbow, 90 deg. standard	1
2, 9, 13, 34, 38	Elbow, 90 deg. standard	2
7, 11, 19, 20, 22, 24, 25, 27, 29, 32, 36, 40, 41, 42, 43, 44, 45, 47, 49, 50, 52	Tee or Cross, (flow turned 90 deg.)	1
15, 16, 17, 18, 40, 41, 42, 43	Elbow, 90 deg. standard and Tee or Cross, (flow turned 90 deg.)	1 Tee and 1 Elbow

Junction Properties

To globally edit the junctions, select Global Junction Edit from the Edit menu. Next, click the appropriate junction based on what you are inputting. For example, Spray Discharge. Click All to select all junctions in the list. Then, click Select Spray Discharge Data and enter the following inputs in the pipe properties window. Repeat this process for all junction types in the model using the inputs below.

1. All Spray Discharges
 - a. Elevation = 3 meters
 - b. Loss Model = K (Fire Sprinkler)
 - c. Exit Properties = Pressure
 - d. Exit Pressure = 0 barG

- e. K (Fire Sprinkler) = 3 liters/min per square root of bar
- f. Number of Sprinklers = 1
- 2. All Branches
 - a. Elevation = 3 meters
- 3. J1 Assigned Pressure
 - a. Elevation = 0.15 meters
 - b. Pressure = 4.41 barG
 - c. Pressure Specification = Static
- 4. J2 Valve
 - 1. Inlet Elevation = 0.76 meters
 - 2. Outlet Elevation = 1.22 meters
 - 3. Valve Data Source = Equivalent Length (Metric)
 - 4. Equivalent Length = User Defined
 - 5. Equivalent Length (User) = 3.96 meters

Import Excel Change Data

After you have input for all pipes and junctions, import the Excel Change data spreadsheet. To do this, import the spreadsheet in the Examples Help file entitled ***Metric - NFPA AFT Transfer Spreadsheet.xlsx*** by clicking on File > Import Excel Change Data, and then browsing to this spreadsheet. After all changes were made, a window will appear that lets you know that all changes were made.

Step 6. Create the NFPA Group

In order to generate an NFPA report, AFT Fathom needs to know the path for which you wish to generate the report. Generally, this report is developed for the hydraulically most remote path. For this model the hydraulically most remote path is either the path to J11 or J15. For Fathom to generate a path it will first need a group containing all pipes and junctions along the desired flow path.

To create a group, select all pipes and junctions that are along the flow path. For this example you will need to select pipes P1, P2, P3, P4, P5, P6, P7, P8, P9, and P10, and junctions J1, J2, J3, J4, J5, J6, J7, J8, J9, J10, and J11 (see this path selected in Figure 2). Note, to select multiple objects, hold the shift key while selecting each with this mouse. Alternatively, you can select these pipes and junctions directly from the Group Manager. Once these pipes and junctions are selected, open the Group Manager by going to Edit > Groups > Manager. Click on Create Group and name the group *NFPA Path*. Make sure this path is selected under Existing Groups, then under both the Pipes in Group and Junctions in Group columns, select Workspace. Press OK to confirm this group.

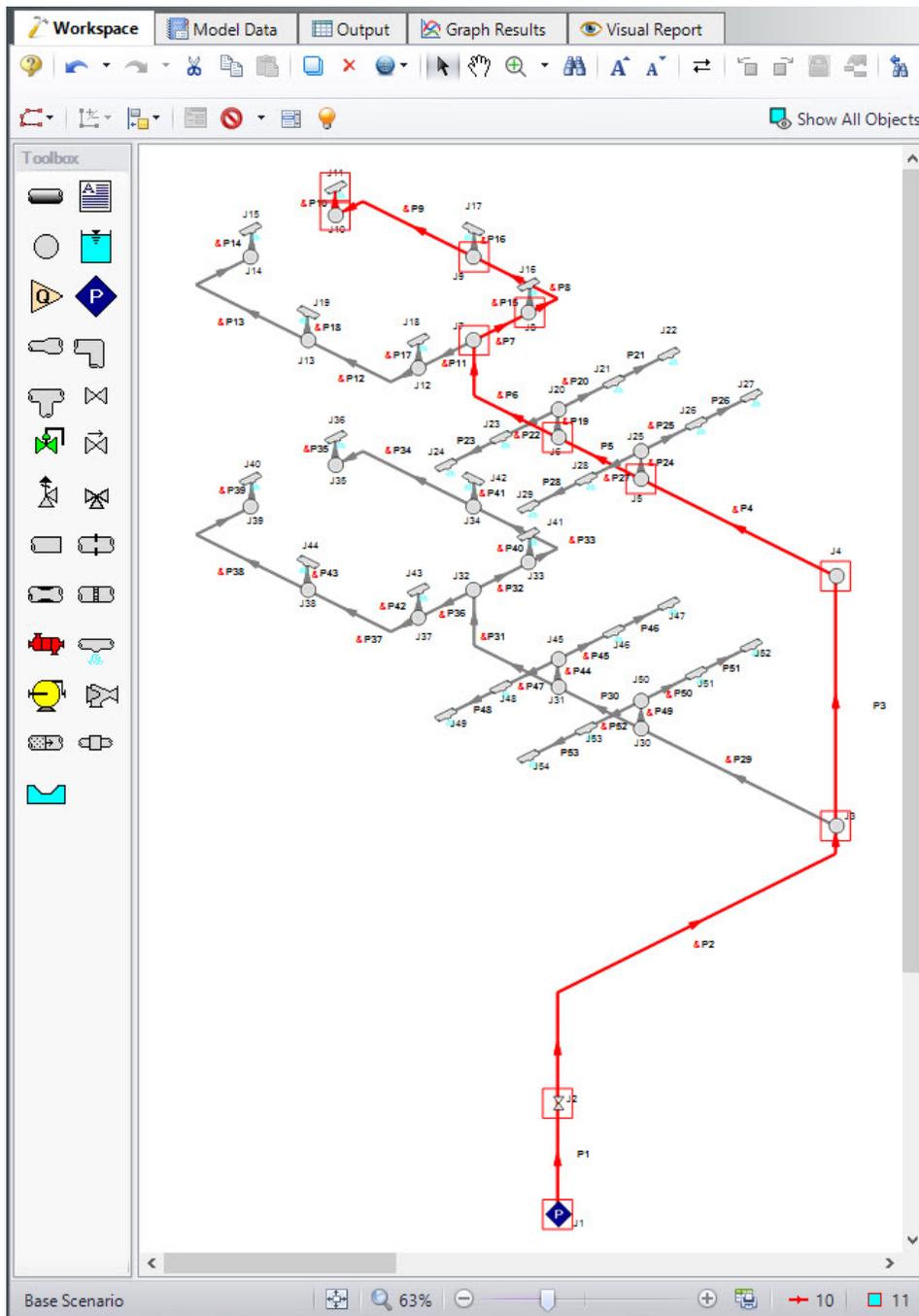


Figure 2: NFPA Group selected on the Workspace

The group we just selected contains the hydraulically most remote path, but this information was simply provided for the sake of convenience in this example and is not frequently known without modeling the system. If you have the GSC module, you can use it to determine the most hydraulically remote path. See the Example problem [NFPA Firewater System - GSC](#) if you are interested in seeing how to use the GSC module for this. Otherwise, if the hydraulically most remote path is not known, you will need to dig through Fathoms output to determine which spray nozzle is receiving the least flow.

Step 7. Specify the NFPA Path

Once you know the hydraulically most remote path, you will generate an NFPA report for this path. To specify which path in AFT Fathom you want to generate the NFPA report for, open Analysis Setup and go to the NFPA panel. Select the group in the Select a Group for the NFPA Report list.

Step 8. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

You will notice there are numerous cautions about the pipe lengths being shorter than the elevation change. This model used pipe lengths and elevations based off a NFPA example. In the NFPA example, the length of the spray discharges raised the outlet height of the junction. Fathom treats junctions as single points in space, so to keep the inputs identical the pipe length had to be a little bit short.

Step 9. Examine the Output

The Output window contains the NFPA Report, as well as the standard AFT Fathom output parameters. To view the NFPA Report, click on the NFPA Report tab in the General section of the Output window. You can expand this section by dragging the border down. The NFPA Report can be printed from File > Print NFPA Report, or you can select all cells by clicking the top-left corner and copy-paste into Excel for modifications. The NFPA Report for this model is shown in Figure 3a and Figure 3b.

NFPA Firewater System

Hydraulic Calculations										
For: _____										Sheet 1 of 2
_____										By: _____
_____										Date: 9/8/2023
_____										Job No.: _____
Step No.	Nozzle Ident. and Location	Flow in L/min	Pipe Size Nominal / ID (mm)	Pipe Fittings and Devices (m)	Equiv. Pipe Length (m)	Friction Loss (bar/m)	Pressure Summary (barG)	Normal Pressure (barG)	Notes	Ref. Step
6					L 0.200		Pt 1.38	Pt 1.38		
7	J11 (Spray Discharge)	q	1 inch (26.6446)	E=0.610	F 0.610		Pe 0.029	Pv 0.103		
8		Q 152.1			T 0.810	0.107	Pf 0.087	Pn 1.27		
9					L 1.80		Pt 1.49	Pt 1.49		
10	J10 (Branch)	q	1 inch (26.6446)	2E=1.22	F 1.22		Pe 0.0	Pv 0.103		
11		Q 152.1			T 3.02	0.107	Pf 0.323	Pn 1.39		
12					L 1.80		Pt 1.82	Pt 1.82		
13	J9 (Branch)	q 159.4	1-1/4 inch (35.052)	E=0.914	F 0.914		Pe 0.0	Pv 0.145		
14		Q 311.5			T 2.71	0.106	Pf 0.288	Pn 1.67		
15					L 0.600		Pt 2.10	Pt 2.10		
16	J8 (Branch)	q 172.1	1-1/2 inch (40.894)	T=2.44	F 2.44		Pe 0.0	Pv 0.188		
17		Q 483.6			T 3.04	0.113	Pf 0.344	Pn 1.92		
18					L 1.50		Pt 2.45	Pt 2.45		
19	J7 (Branch)	q 483.5	2 inch (52.50179)	E=1.52	F 1.52		Pe 0.029	Pv 0.277		
20		Q 967.1			T 3.02	0.121	Pf 0.366	Pn 2.17		
21					L 1.20		Pt 2.84	Pt 2.84		
22	J6 (Branch)	q 756.9	2-1/2 inch (62.7126)		F 0.0		Pe 0.0	Pv 0.432		
23		Q 1724.0			T 1.20	0.149	Pf 0.178	Pn 2.41		
24					L 1.50		Pt 3.02	Pt 3.02		
25	J5 (Branch)	q 781	3 inch (77.92719)	E=2.13	F 2.13		Pe 0.0	Pv 0.383		
26		Q 2505.0			T 3.63	0.103	Pf 0.374	Pn 2.64		
27					L 3.00		Pt 3.40	Pt 3.40		
28	J4 (Branch)	q	3 inch (77.92719)		F 0.0		Pe 0.303	Pv 0.383		
29		Q 2505.0			T 3.00	0.103	Pf 0.309	Pn 3.01		

Figure 3a: NFPA Report Page 1

Hydraulic Calculations										
For: _____										Sheet 2 of 2
_____										By: _____
_____										Date: 9/8/2023
_____										Job No.: _____
Step No.	Nozzle Ident. and Location	Flow in L/min	Pipe Size Nominal / ID (mm)	Pipe Fittings and Devices (m)	Equiv. Pipe Length (m)	Friction Loss (bar/m)	Pressure Summary (barG)	Normal Pressure (barG)	Notes	Ref. Step
34	J3 (Branch)	q 2630.6	6 inch (154.051)	V=3.96	L 5.20		Pt 4.01	Pt 4.01		
35		Q 5135.6		2E=8.53	F 12.5		Pe 0.190	Pv 0.105		
36					T 17.7	0.014	Pf 0.295	Pn 3.90		
37					L 0.600		Pt 4.45	Pt 4.45		
38	J2 (Valve)	q	6 inch (154.051)		F 0.0		Pe 0.060	Pv 0.105		
39		Q 5135.6			T 0.600	0.014	Pf 0.008	Pn 4.34		

Figure 3b: NFPA Report Page 2

Conclusion

AFT Fathom allows you to display the hydraulic calculations for an NFPA report. In this example, you have built a model of a firewater system and generated an NFPA hydraulic report. In this report we can see that the minimum pressure for this system is 4.41 barG in order to see 152 L/min at the most hydraulically remote nozzle.

NFPA Firewater System

To those familiar with the NFPA Example this was based off, you may notice a slight difference in the minimum pressures calculated by Fathom. This is because Fathom uses a more detailed model to account for various behavior in the system. You can read more about these differences on the [NFPA Panel](#) topic.

Crude Oil Pipeline

Summary

The objective of this example is to model a pipeline system to investigate the effects of a viscous fluid. We will answer a number of questions about the pipeline system such as:

1. How does viscosity affect pump performance?
2. What is the flow rate?
3. What is the maximum pressure in the pipe?
4. Is the flow laminar or turbulent?
5. How high can the hill be until a vacuum (i.e., less than atmospheric pressure) is created?
6. How does the hill height affect flow rate?
7. How much can the pump speed be slowed down and still not pull a vacuum?

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Crude Oil Pipeline.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:

- a. Fluid Library = User Specified Fluid
- b. Name = Petroleum Product
- c. Density = 0.8 S.G. water
- d. Dynamic Viscosity = 500 centipoise

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

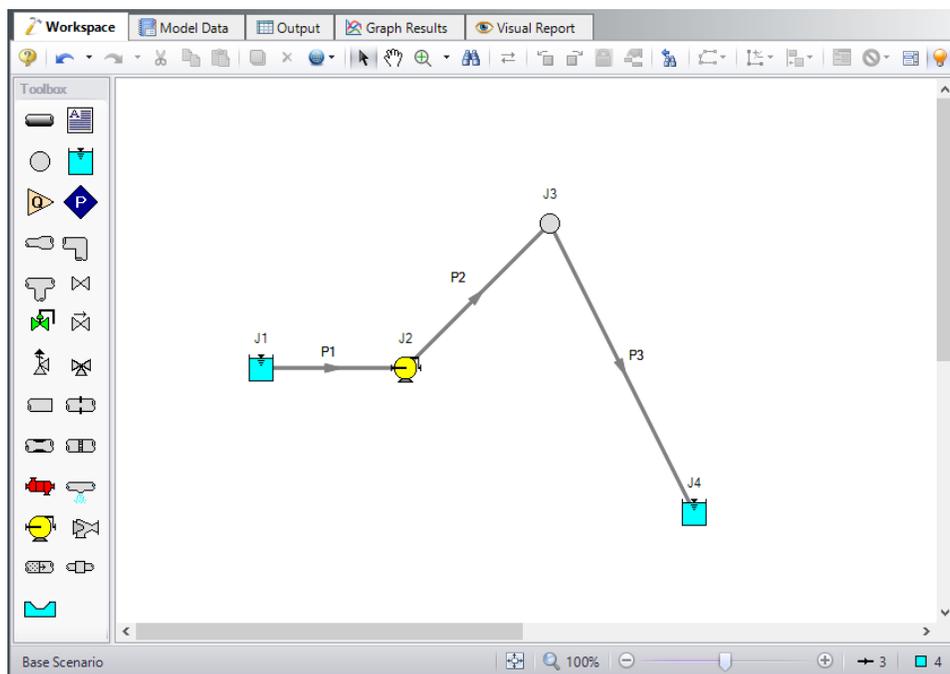


Figure 1: Model for Pipeline Problem

Pipe Properties

1. Pipe Model tab
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 24 inch
 - d. Type = STD (schedule 20)
 - e. Friction Model Data Set = Standard
 - f. Lengths =

Pipe	Length
1	30 meters
2	40 km
3	80 km

Junction Properties

Enter the properties of the junctions in the workspace as follows:

1. J1 Reservoir
 - a. Liquid Surface Elevation = 230 meters
 - b. Liquid Surface Pressure = 0 barG (0 kPa(g))
 - c. Pipe Depth = 3 meters
2. J2 Pump
 - a. Inlet Elevation = 220 meters
 - b. Pump Model = Centrifugal (Rotodynamic)
 - c. Analysis Type = Pump Curve
 - d. Enter Curve Data =

Volumetric	Head
barrels/day	meters
0	610
100,000	550
200,000	425

- e. Curve Fit Order = 2

Note: If barrels/day is not an available unit, select User Options from the Tools menu, click on Preferred Units, and select the radio button for All Units.

4. J3 Branch
 - a. Elevation = 381 meters
5. J4 Reservoir
 - a. Liquid Surface Elevation = 3 meters
 - b. Liquid Surface Pressure = 0 barG
 - c. Pipe Depth = 3 meters

In the Pump Properties window on the Optional tab, Fathom includes an option to Use Viscosity Correction. We will turn this option on later to determine how much this option affects the answers. Leave the option off for now.

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

Open the Output Control window by selecting Output Control from the toolbar or Tools menu. On the Display Parameters tab, select the Pumps button to display the list of output parameters that are specific to pumps. Then add the three correction parameters to pump efficiency CE, pump head rise CH, and pump flow CQ. These are the pump viscosity correction/pump de-rating factors that will de-rate the pump according to the Hydraulic Institute method, ANSI/HI Standard 9.6.7-2010. Change the Volumetric Flow Rate units to barrels/day.

Now select the Pipes button to display the pipe output parameters, then change the units for Volumetric Flow Rate to barrels/day. Also, add Reynolds Number to the list of output variables. Click OK to save the changes and close Output Control.

The Output window contains all the data that was specified in the output control window. The output from this model is shown in Figure 2. Note that the flow rate in all the pipes is 152,565 barrels/day.

The screenshot shows the 'Output' window with three data tables:

Jct	Results Diagram	Name	Vol. Flow (barrels/day)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
2	Show	Pump	152,565	224.4	38.60	492.4	N/A	100.0	1,083	N/A	N/A	N/A	N/A

Pipe	Name	Vol. Flow Rate (barrels/day)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Reynolds No.
1	Pipe	152,565	1.025	1.779	1.244	227.0	220.0	-0.5346	-0.5346	-0.5487	0.1800	1.244	1.779	1.248	1.783	9.676E+02
2	Pipe	152,565	1.025	40.376	8.947	220.0	381.0	31.4289	31.4289	12.6203	239.9464	40.376	8.947	40.380	8.952	9.676E+02
3	Pipe	152,565	1.025	8.947	1.244	381.0	0.0	7.7031	7.7031	-29.8653	479.2706	8.947	1.244	8.952	1.248	9.676E+02

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Reservoir	1.013	1.248	1.013	1.248	1.011	224.4	0
2	Pump	1.779	40.376	1.783	40.380	1.011	224.4	0
3	Branch	8.947	8.947	8.952	8.952	1.011	224.4	0
4	Reservoir	1.013	1.248	1.013	1.248	1.011	224.4	0

Figure 2: Output Window for Pipeline example

Return to the workspace and open the pump properties window. Navigate to the Optional tab and check the box for Use Viscosity Correction. Input the following:

Crude Oil Pipeline

1. Head Correction (ANSI/HI 9.6.7-2015) = Assume Calculated Flow Is At BEP
2. Rated Speed (rpm) = 3600

Click OK and run the model again. The output from this run is shown below. Now notice that the flow rate is 144,781 barrels/day. Click the Pump Summary tab in the General Output section to see the values for CE, CH, and CQ. The output from this model is shown in Figure 3.

The screenshot shows the 'Output' window with three main sections: Pump Summary, Pipes, and All Junctions.

Pump Summary Table:

Jct	Results Diagram	Name	Vol. Flow Rate (barrels/day)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (KW)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)	CE	CH	CQ
2	Show	Pump	144,781	212.9	35.72	455.7	N/A	100.0	951.1	N/A	N/A	N/A	N/A	0.7238	0.9322	0.9322

Pipes Table:

Pipe	Name	Vol. Flow Rate (barrels/day)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Reynolds No.
1	Pipe	144,781	0.9726	1.780	1.245	227.0	220.0	-0.5353	-0.5353	-0.5487	0.1708	1.245	1.780	1.248	1.784	9.182E+02
2	Pipe	144,781	0.9726	37.503	7.034	220.0	381.0	30.4692	30.4692	12.6203	227.7041	37.503	7.034	37.507	7.038	9.182E+02
3	Pipe	144,781	0.9726	7.034	1.245	381.0	0.0	5.7896	5.7896	-29.8653	454.8593	7.034	1.245	7.038	1.248	9.182E+02

All Junctions Table:

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Reservoir	1.013	1.248	1.013	1.248	959.1	212.9	0
2	Pump	1.780	37.503	1.784	37.507	959.1	212.9	0
3	Branch	7.034	7.034	7.038	7.038	959.1	212.9	0
4	Reservoir	1.013	1.248	1.013	1.248	959.1	212.9	0

Figure 3: Output Window for Pipeline example with viscosity correction

Question 1. How does viscosity affect pump performance?

The flow rate, pump head, and overall power all decreased when the viscosity corrections were used.

Question 2. What is the flow rate?

Before the viscosity correction was added the flow rate was 152,565 barrels/day. After the viscosity correction was added the flow rate became 144,781 barrels/day. Therefore, the viscosity correction factor reduced the flow rate by approximately 5.1%.

Question 3. What is the maximum pressure in the pipe?

The maximum pressure can be found in the pipe summary in the P Static Max column. It occurs in Pipe 2 and it occurs at the inlet of the pipe (outlet of the pump). The pressure is 37.503 bar (3,750.3 kPa) or 36.4902 barG (3,649.02 kPa (g)).

Question 4. Is the flow laminar or turbulent?

In the pipe summary window we can see that the Reynolds number is 918.2. Therefore the flow is laminar. The general cut-off for laminar flow is Reynolds number greater than 2,300.

Question 5. How high can the hill be until a vacuum is created?

To answer this question we have to change the elevation of branch J3 and re-run the model. If the hill height is raised to 457.8 meters, the minimum pressure in pipe 2 and 3 and at junction J3 is 1.014 bar (101.4 kPa), or nearly atmospheric pressure. If the hill is higher than 457.8 meters, a vacuum would be created. This output can be seen in Figure 4.

Jct	Results Diagram	Name	Vol. Flow Rate (barrels/day)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (KW)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)	CE	CH	CQ
2	Show	Pump	144.781	212.9	35.72	455.7	N/A	100.0	951.1	N/A	N/A	N/A	N/A	0.7238	0.9322	0.9322

Pipe	Name	Vol. Flow Rate (barrels/day)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Reynolds No.
1	Pipe	144.781	0.9726	1.780	1.245	227.0	220.0	-0.5353	-0.5353	-0.5487	0.1708	1.245	1.780	1.248	1.784	9.182E+02
2	Pipe	144.781	0.9726	37.503	1.014	220.0	457.8	36.4893	36.4893	18.6404	227.7041	37.603	1.014	37.507	1.018	9.182E+02
3	Pipe	144.781	0.9726	1.245	1.014	457.8	0.0	-0.2305	-0.2305	-35.8854	454.8593	1.014	1.245	1.018	1.248	9.182E+02

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Reservoir	1.013	1.248	1.013	1.248	959.1	212.9	0
2	Pump	1.780	37.503	1.784	37.507	959.1	212.9	0
3	Branch	1.014	1.014	1.018	1.018	959.1	212.9	0
4	Reservoir	1.013	1.248	1.013	1.248	959.1	212.9	0

Figure 4: Output Window for Pipeline example with 457.8 meter hill

Question 6. How does the hill height affect flow rate?

Compare Figure 3 and Figure 4 and you will see that the flow rate is exactly the same. Because the pipe length is the same and the net elevation change is the same, the flow rate will be the same.

Question 7. How much can the pump speed be slowed down and still not pull a vacuum?

The pump can be slowed down to 86.6% before the minimum pressure in pipes 2 and 3 goes below atmospheric (1.013 bar) (101.3 kPa). The output of this run is shown in Figure 5.

Crude Oil Pipeline

Workspace | Model Data | **Output** | Graph Results | Visual Report

General | Warnings | Design Alerts | Applied Standards | Pump Summary | Reservoir Summary

Jct	Results Diagram	Name	Vol. Flow Rate (barrels/day)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)	CE	CH	CQ
2	Show	Pump	120.619	177.4	26.79	341.8	N/A	86.60	594.2	N/A	N/A	N/A	N/A	0.6988	0.9220	0.9220

Pipes

Pipe	Name	Vol. Flow Rate (barrels/day)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Reynolds No.
1	Pipe	120.619	0.8103	1.783	1.246	227.0	220.0	-0.5376	-0.5376	-0.5487	0.1423	1.246	1.783	1.248	1.786	7.650E+02
2	Pipe	120.619	0.8103	28.574	1.084	220.0	381.0	27.4905	27.4905	12.6203	189.7032	28.574	1.084	28.577	1.086	7.650E+02
3	Pipe	120.619	0.8103	1.246	1.084	381.0	0.0	-0.1622	-0.1622	-29.8653	378.9309	1.084	1.246	1.086	1.248	7.650E+02

All Junctions | Branch | Pump | Reservoir

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Reservoir	1.013	1.248	1.013	1.248	799.0	177.4	0
2	Pump	1.783	28.574	1.786	28.577	799.0	177.4	0
3	Branch	1.084	1.084	1.086	1.086	799.0	177.4	0
4	Reservoir	1.013	1.248	1.013	1.248	799.0	177.4	0

Base Scenario | 100%

Figure 5: Output Window for Pipeline example with pump operating at 86.6% speed

Hot Water System

Summary

The objective of this example is to select the best pump configuration for a hot water system. The design goals of the system are:

- Minimum 0.42 bar (42 kPa) drop across flow control valves
- Maximum 3 meters/sec velocity in all pipes
- No NPSH violations
- All pumps must operate between 70% and 100% of BEP
- The selected pump must also work in a special operating mode with one pump off
- The best pump will meet all the requirements and use the least power.

Topics Covered

- Using Junction Libraries to enter data
- Using multiple pump configurations
- Using NPSH and Efficiency data for pumps
- Turning off parts of a model to simulate different operating conditions

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Hot Water System.fth***
- ***Hot Water System.dat*** - engineering library

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

Hot Water System

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = AFT Standard
 - b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
 - c. Temperature = 65 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

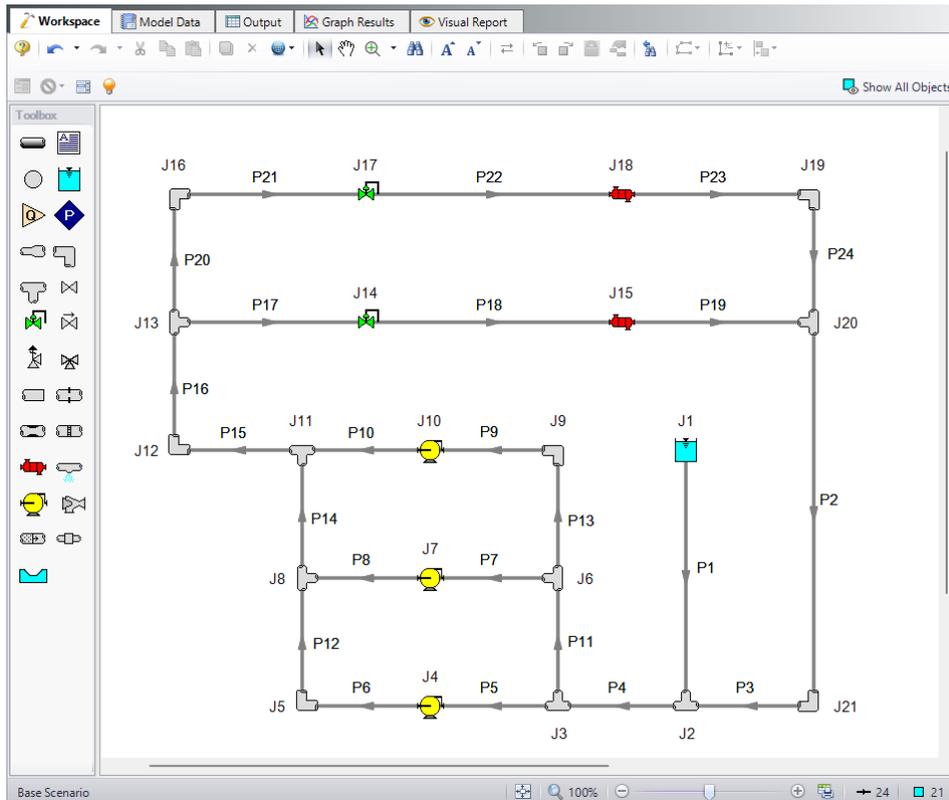


Figure 1: Model Layout for Hot Water System

Note: A safety factor of 10% will be added to all pipe and junction friction losses. This is added by typing in 1.1 in the Pipe Friction field for Design Factors found on the Optional tab in the Pipe Properties window. The Design Factor of 1.1 will be added to the Junction Friction Losses on the Optional tab in the Junction Properties windows for every junction except for the three pumps.

Pipe Properties

1. Pipe Model tab
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = Use table below
 - d. Type = STD (schedule 40)
 - e. Friction Model Data Set = Standard
 - f. Lengths = Use table below

Pipe	Size	Length (meters)
12, 13	4 inch	10
6, 8, 10, 18, 20, 22, 24	4 inch	15
5, 7, 9, 17, 21	4 inch	30
19, 23	4 inch	50
11, 14	6 inch	10
3, 15	6 inch	15
1, 4, 16	6 inch	30
2	6 inch	90

2. Optional tab
 - a. Design Factors
 - i. Pipe Friction = 1.1

Junction Properties

1. J1 Reservoir
 - a. Name = Topping Tank
 - b. Liquid Surface Elevation = 3 meters
 - c. Liquid Surface Pressure = 0 barG (0 kPa (g))
 - d. Pipe Depth = 3 meters
2. All Tees

Hot Water System

- a. Elevation = 0 meters
- b. Loss Model = Simple
3. All Bends
 - a. Inlet Elevation = 0 meters
 - b. Type = Standard Elbow (knee, threaded)
 - c. Angle = 90 Degrees
4. J14 & J17 Control Valves
 - a. J14 Name = FCV A
 - b. J17 Name = FCV B
 - c. Inlet Elevation = 0 meters
 - d. Valve Type = Flow Control (FCV)
 - e. Control Setpoint = Volumetric
 - f. Flow Setpoint = 50 m³/hr
5. All Pumps & Heat Exchangers
 - a. J4 Name = Pump A
 - b. J7 Name = Pump B
 - c. J10 Name = Pump C
 - d. J15 Name = Heat Exchanger A
 - e. J18 Name = Heat Exchanger B
 - f. Inlet Elevation = 0 meters

To show the names of the pumps, control valves, heat exchangers, and reservoirs, a new Workspace Layer will be created.

1. Create a new layer by selecting New Layer and then Blank Layer from the Workspace Layers tab of the Quick Access Panel.
2. Name the layer Junction Labels.
3. Edit the layer by selecting it and clicking the gear icon to bring up the Layer Settings window.
4. Open the Show/Hide Labels panel and uncheck Force Shown Labels to Match Shown Objects.
5. Toggle the visibility icon next to the pumps, heat exchangers, control valves, and reservoir to on. This will cause their labels to be shown.
6. Navigate to the Junction Parameters panel and expand the Commonly Used Junction Parameters list.
7. Double-click Junction Name to add it to the list on the right-hand side. This will add the Junction Name to the label of the junctions that were defined in the Show/Hide Labels panel.
8. Close the Layer Settings window.

To enter the pump and heat exchanger data, we will use a pre-defined library:

1. Open the Library menu and select the Library Manager.
2. At the bottom, click Add Existing Library. Navigate to the AFT Fathom 13 Examples folder and select the file titled **Hot Water System.dat** and Open. The library *Hot Water System* will be added to the list of available libraries.

Note: Typically when a Library is first added to the list in Library Manager, it will automatically be connected and will appear with a checkmark next to it. Step 3 below is only needed if the library is not automatically connected.

3. If not already, check the box next to the library name Hot Water System, this will connect the library to the model.
4. Close Library Manager
5. Now open the Pump Properties window for Pump A, and in the Library Jct field at the top of the properties window, select Metric - Hot Water Pump with Multiple Configs from the drop down menu.
6. In the Pump Curve area, make sure RPM = 1800 RPM and Impeller = 6 inches.
7. Repeat this process for Pump B and Pump C.
8. Open the Heat Exchanger Properties Window for Heat Exchanger A, and select Metric - Hot Water Heat Exchanger in the library drop down box. This library includes the pressure loss data for the heat exchangers.
9. Repeat these steps for Heat Exchanger B.

Finally, use Global Junction Edit to add the 10% safety factor.

1. Select Global Edit then Global Junction Edit from either the toolbar or Edit menu.
2. Select Common Data
3. Click Select Common Junction Data
4. In the Design Factor area, set Junction Friction Loss = 1.1
5. Click OK
6. Click All to select all junctions in the list
7. Check the box in the Parameters to Change (Select in List) area next to Design Factor 1.1
8. Click Apply Selections. A message should appear to inform you that changes were successively applied.
9. Click OK.

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

After running the model, select the Pump Summary tab and review these results. The pump summary data is shown in Figure 2.

Hot Water System

Pump Summary													
Jct	Results Diagram	Name	Vol. Flow (m3/hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
4	Show	Pump A	32.90	8.971	1.125	11.69	78.66	100.0	1.306	35.93	91.59	10.164	0.6632
7	Show	Pump B	34.21	9.328	1.120	11.63	78.98	100.0	1.346	35.93	95.23	10.074	0.6824
10	Show	Pump C	32.90	8.971	1.125	11.69	78.66	100.0	1.306	35.93	91.59	9.945	0.6632

Figure 2: Pump Summary for First Configuration

Now we must review these results to determine if the requirements are met. The pumps operate at 91% to 96% of BEP and there are no NPSH violations. Although the pump meets the requirements, we can see from the valve summary in Figure 3 that the flow control valves do not maintain a minimum of 0.7 bar (70 kPa) pressure drop. The velocity in all the pipes is less than 3 meters/sec. The pipe summary is shown in Figure 4.

Valve Summary										
Jct	Name	Valve Type	Vol. Flow (m3/hr)	Mass Flow (kg/sec)	dP Stag. (bar)	dH (meters)	P Static In (bar)	Cv	K	Valve State
14	FCV A	FCV	50.00	13.63	0.2627	2.729	2.159	111.8	18.72	Open
17	FCV B	FCV	50.00	13.63	0.1674	1.740	2.111	140.0	11.93	Open

Figure 3: Valve Summary for First Configuration

Pipes															
Pipe	Name	Vol. Flow Rate (m3/hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)
1	Pipe	0.00	0.000	1.302	1.302	0	0	0.000000	0.000000	0	0.00000	1.302	1.302	1.302	1.302
2	Pipe	100.00	1.490	1.429	1.315	0	0	0.113448	0.113448	0	1.17868	1.429	1.315	1.440	1.326
3	Pipe	100.00	1.490	1.310	1.291	0	0	0.018908	0.018908	0	0.19645	1.310	1.291	1.321	1.302
4	Pipe	100.00	1.490	1.291	1.253	0	0	0.037816	0.037816	0	0.39289	1.291	1.253	1.302	1.264
5	Pipe	32.90	1.113	1.258	1.223	0	0	0.035526	0.035526	0	0.36910	1.258	1.223	1.264	1.229
6	Pipe	32.90	1.113	2.347	2.330	0	0	0.017763	0.017763	0	0.18455	2.347	2.330	2.354	2.336
7	Pipe	34.21	1.157	1.252	1.213	0	0	0.038284	0.038284	0	0.39775	1.252	1.213	1.258	1.220
8	Pipe	34.21	1.157	2.333	2.314	0	0	0.019125	0.019125	0	0.19871	2.333	2.314	2.340	2.321
9	Pipe	32.90	1.113	1.237	1.201	0	0	0.035526	0.035526	0	0.36910	1.237	1.201	1.243	1.208
10	Pipe	32.90	1.113	2.326	2.309	0	0	0.017738	0.017738	0	0.18429	2.326	2.309	2.332	2.315
11	Pipe	67.13	1.000	1.259	1.253	0	0	0.005856	0.005856	0	0.06084	1.259	1.253	1.264	1.258
12	Pipe	32.90	1.113	2.326	2.315	0	0	0.011817	0.011817	0	0.12278	2.326	2.315	2.332	2.321
13	Pipe	32.90	1.113	1.252	1.240	0	0	0.011842	0.011842	0	0.12303	1.252	1.240	1.258	1.246
14	Pipe	67.13	1.000	2.316	2.310	0	0	0.005856	0.005856	0	0.06084	2.316	2.310	2.321	2.315
15	Pipe	100.00	1.490	2.304	2.285	0	0	0.018908	0.018908	0	0.19645	2.304	2.285	2.315	2.296
16	Pipe	100.00	1.490	2.280	2.242	0	0	0.037816	0.037816	0	0.39289	2.280	2.242	2.290	2.253
17	Pipe	50.00	1.691	2.239	2.159	0	0	0.079538	0.079538	0	0.82637	2.239	2.159	2.253	2.173
18	Pipe	50.00	1.691	1.896	1.857	0	0	0.039769	0.039769	0	0.41318	1.896	1.857	1.910	1.871
19	Pipe	50.00	1.691	1.558	1.426	0	0	0.132563	0.132563	0	1.37728	1.558	1.426	1.572	1.440
20	Pipe	50.00	1.691	2.239	2.199	0	0	0.039769	0.039769	0	0.41318	2.239	2.199	2.253	2.213
21	Pipe	50.00	1.691	2.191	2.111	0	0	0.079538	0.079538	0	0.82637	2.191	2.111	2.205	2.125
22	Pipe	50.00	1.691	1.944	1.904	0	0	0.039769	0.039769	0	0.41318	1.944	1.904	1.958	1.918
23	Pipe	50.00	1.691	1.606	1.473	0	0	0.132563	0.132563	0	1.37728	1.606	1.473	1.620	1.487
24	Pipe	50.00	1.691	1.465	1.426	0	0	0.039769	0.039769	0	0.41318	1.465	1.426	1.480	1.440

Figure 4: Pipe Output for First Configuration

After examining this pump configuration, this process needs to be repeated for each configuration, in order to find which pump best meets the specified requirements. A tool that can be used for this kind of evaluation is the Scenario Manager. Open the Scenario Manager on the Quick Access Panel and create five scenarios, one for each pump configuration. Below each scenario, create a child scenario for each

Hot Water System

configuration named Pump A Off. The Scenario Manager on the Quick Access Panel should look like Figure 5. Alternatively, you can access the Scenario Manager from the Tools menu.

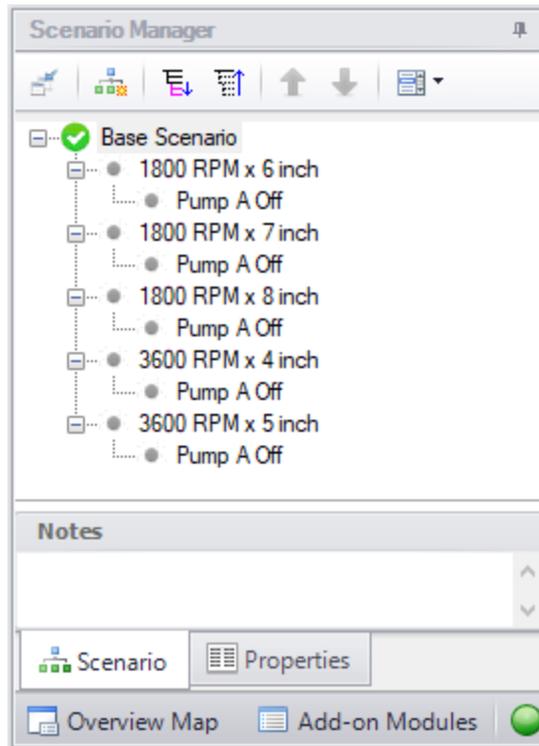


Figure 5: Scenario Manager on the Quick Access Panel for Hot Water System

Now, work your way through each scenario and change the pump configuration to the pump configuration for each scenario. For the scenarios where Pump A is off, use the Special Condition Pump Off No Flow. Run each scenario, note the sum of the three pump's power usage, and determine if the pump meets all the requirements. It would help to create a table of the requirements and configurations as shown in Table 1.

Table 1: Summary Table for Hot Water System

Pump	1800 x 6	1800 x 7	1800 x 8	3600 x 4	3600 x 5
Normal Mode					
BEP OK?	Yes	Yes	No	No	No
NPSH OK?	Yes	Yes	Yes	Yes	Yes
FCV dP OK?	No	Yes	Yes	Yes	Yes
Pipe Velocity OK?	Yes	Yes	Yes	Yes	Yes

Hot Water System

Pump	1800 x 6	1800 x 7	1800 x 8	3600 x 4	3600 x 5
Power Used (kW)	3.958	5.643	7.770	7.770	13.504
Special Mode (Pump A Off)					
NSPH OK?	Yes	Yes	Yes	Yes	Yes
FCV dP OK?	No	Yes	Yes	Yes	Yes
Pipe Velocity OK?	Yes	Yes	Yes	Yes	Yes

From Table 1, it can be seen that the only configurations that meet all the requirements are the 1800 RPM x 7-inch impeller. One thing to note is that the 1800 RPM x 8-inch and 3600 RPM x 4-inch configurations are actually the *same* pump configuration due to the affinity laws. The power usage of both configurations help show that they are the same pump.

Note: The 1800 RPM x 6-inch configuration with Pump A turned off is a case where the flow control valves will have to add pressure in order to maintain their set point. A flow control valve adding pressure is physically unrealistic, however, it is useful for troubleshooting purposes because it shows how close a control valve is to being able to maintain set point. The fact that the FCVs fail with this particular scenario is an example of how a certain pump configuration and operating condition may not be able to provide the system pressure and meet the requirements.

Plant Cooling Cost Calculation

Summary

This example demonstrates the fundamental concepts of doing a cost analysis with AFT Fathom. The example illustrates how to calculate initial and life cycle costs for a given system design.

After designing a plant cooling system, it is necessary to calculate the system cost over a 10-year period to determine the feasibility of the design. The system model consists of four circulating water pumps, schedule 20 steel - ANSI pipes and fittings, and two sets of cooling tower cells.

To determine the cost of the cooling system for a 10-year period, include material, installation, and energy costs for the pumps, pipes, and fittings. The cooling towers costs will be determined apart from the system cost (cooling tower cost calculations are not included in this example).

Topics Covered

- Creating cost libraries
- Entering pipe, junction and fitting cost data into libraries
- Connecting cost libraries
- Using Cost Settings
- Using the Cost Report

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

In addition, it is assumed that the user has worked through the [Beginner - Cost Calculation](#) example and is familiar with the basics of doing cost calculations.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Plant Cooling Cost Calculation - Initial.fth***
- ***Metric - Plant Cooling Cost Calculation - Final.fth***
- ***Plant Cooling.dat*** - existing engineering library
- ***Plant Cooling - Pump Costs.cst*** - completed cost library for ***Plant Cooling.dat***

Plant Cooling Cost Calculation

- **Plant Cooling - Elbow Costs.cst** - completed cost library for **AFT INTERNAL LIBRARY**
- **Plant Cooling - Pipe Costs Metric.cst** - completed cost library for **Steel - ANSI**

Note: The cost libraries above will be recreated as part of this example, but they are also included as reference material.

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **Metric - Plant Cooling Cost Calculation - Initial.fth** model file listed above, which is located in the Examples folder in the AFT Fathom application folder. **Save the file to a different folder.**

The model should appear as shown in Figure 1.

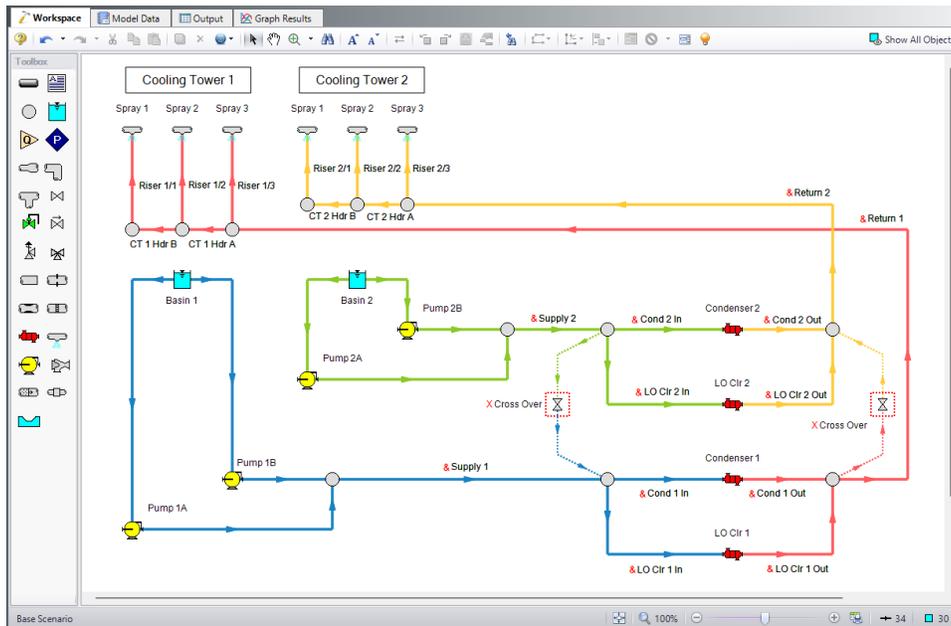


Figure 1: The Plant Cooling Cost Calculation model

Step 3. Define the Cost Settings group

By default, the cost calculations are turned off. To turn on the cost calculations, open Analysis Setup, navigate to the Cost Settings panel, and select Calculate in the Cost Calculation section. The Cost Settings

panel is shown in Figure 2.

The Cost Definitions section allows you to specify the type of costs to be determined.

In the Cost Definitions area, select Material, Installation, and Operation/Energy costs by checking the box next to the name in the list.

In the Energy Cost section, select Use This Energy Cost Information, and enter a Cost of 0.12 U.S. Dollars Per kW-hr.

In the Cost Time Period section, enter a System Life of 10 years.

The Cost Settings panel should look like Figure 2. Close the Analysis Setup window by clicking OK.

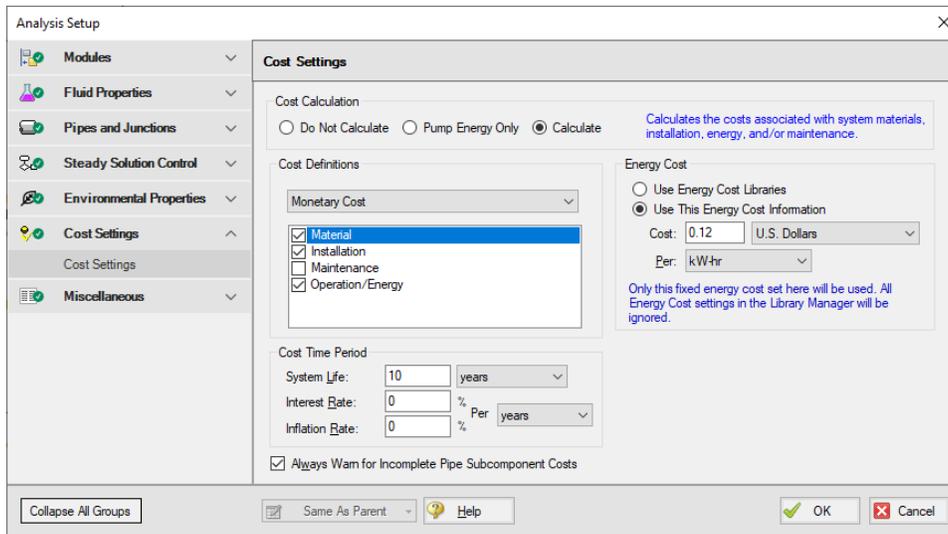


Figure 2: The cost calculations are selected on the Cost Settings window

Step 4. Create the Cost Libraries

Refer to the AFT Fathom Help Content for detailed information regarding creating and using cost libraries.

The energy cost for the pumps was specified in the Cost Settings as a fixed cost rate. Now, the material and installation cost for the pumps, pipes, and fittings must be included. This will be done by creating **three new cost libraries**.

The **first** cost library will be for the pipe material and installation costs and it will be associated with the engineering library that contains the pipe material data, which is the Steel - ANSI pipe material library.

The **second** cost library will be for the additional pipe fittings and losses costs for the elbow losses that are lumped into the pipes. The second cost library will be associated with the AFT INTERNAL LIBRARY. This is because the AFT Internal Library contains the loss information for the additional fittings and losses tab in the Pipe Properties window.

The **third** cost library will be for the pump material and installation costs, and it will be associated with the engineering library that contains the pump information (i.e., the **Plant Cooling.dat** engineering library).

Create a cost library for the pipe material and installation costs

Create a new cost library for the pipe costs by opening the Library menu and selecting Cost Library. Create a new cost library by clicking New at the bottom. When you click New, you are prompted to choose the engineering library with which the cost library will be associated.

The piping material data used in the model comes from the default Steel - ANSI Pipe Material Library, so the pipe cost library will be connected to this library, as well. Select **Steel - ANSI** from the library list shown in the Select Library window, then click Select.

Important: Make this choice carefully, because once you have made the association you cannot change it.

Once the library is created, you must enter a meaningful description and select the cost units. Enter the following data on the General tab:

1. Cost Type = Monetary
2. Monetary Unit = U.S. Dollars
3. Description = Plant Cooling - Pipe Costs Metric
4. Notes = This library is for the Plant Cooling Cost Calculation example file

Save the cost library to a file by clicking Save. Specify a file name such as, **Plant Cooling - Pipe Costs Metric.cst**, and click Save. After saving, if prompted, choose to add the new library to the list of available and connected libraries by clicking Yes.

The information on the General tab should appear as shown in Figure 3. The selected library file information, cost type and monetary units, a description of the cost library, and any descriptive notes you have added will be displayed.

Be sure to keep the Edit Cost Library window open for the next step. For this cost library, you will enter pipe material and installation costs on the Pipe Material tab.

Note: The library filenames may be different than those shown in Figure 3.

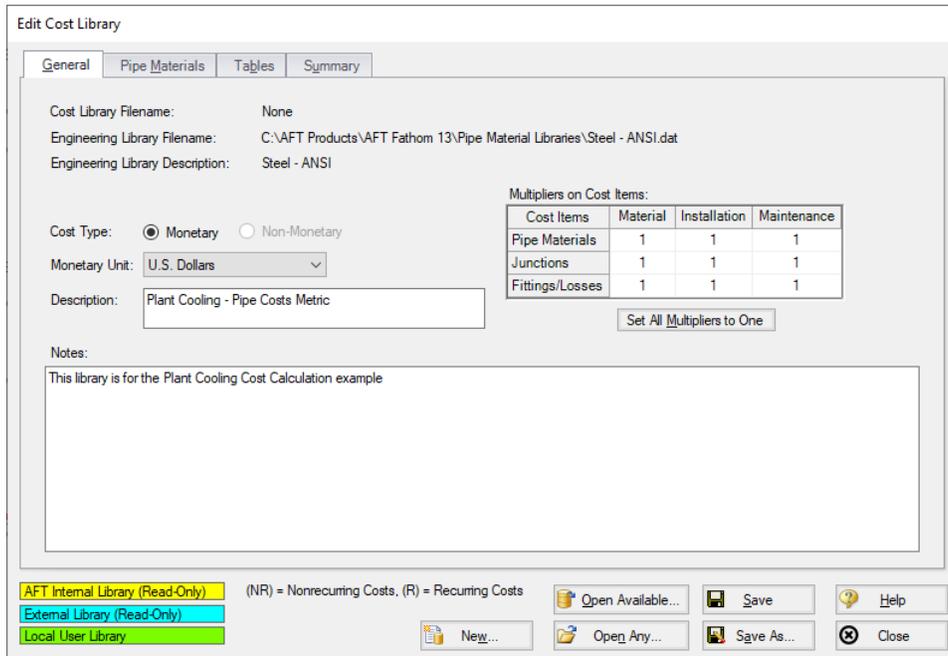


Figure 3: The Edit Cost Library window is used to enter cost library data

Enter the pipe material costs

After the pipe cost library has been created, select the Pipe Materials tab. This is where you will enter the material and installation costs for the piping.

The Pipe Materials tab shows all the pipes in the engineering library (AFT Steel - ANSI Pipe Material Library). Costs can be entered at several levels. You can enter costs at the material level, the nominal size level, and finally at the type (i.e., schedule) level. Costs entered at the material level apply to all nominal sizes and types in that material type. Costs entered at the nominal size level apply to all schedules within that nominal size. Costs entered at the type (schedule) level apply only to that type. **For this example, all of the pipe material costs will be entered at the type (schedule) level for specific sizes.**

To enter a new cost, start by switching the Sort selection to Material, Type or Schedule, Nominal Size. Navigate to Steel - ANSI, STD, 8 inch. Next click New Cost to create a new cost item in the table. The new cost item will appear as a new column.

Each pipe size in this example has two non-recurring (NR) costs associated with it. The costs to be entered are the material and installation costs. You will need to make a second column and change Cost Type to Installation (NR). Figure 4 shows the costs entered for 8-inch STD Steel - ANSI.

Enter the non-recurring pipe costs for the pipe in this example, as shown below.

All pipes are STD Steel - ANSI:

Plant Cooling Cost Calculation

Nominal Size (inches)	Material Cost (dollars/meter)	Installation Cost (dollars/meter)
8	34.50	78.75
14	74.00	169.00
18	90.00	229.75
20	116.25	280.50
24	131.25	316.75
28	190.35	361.00
30	197.00	423.25

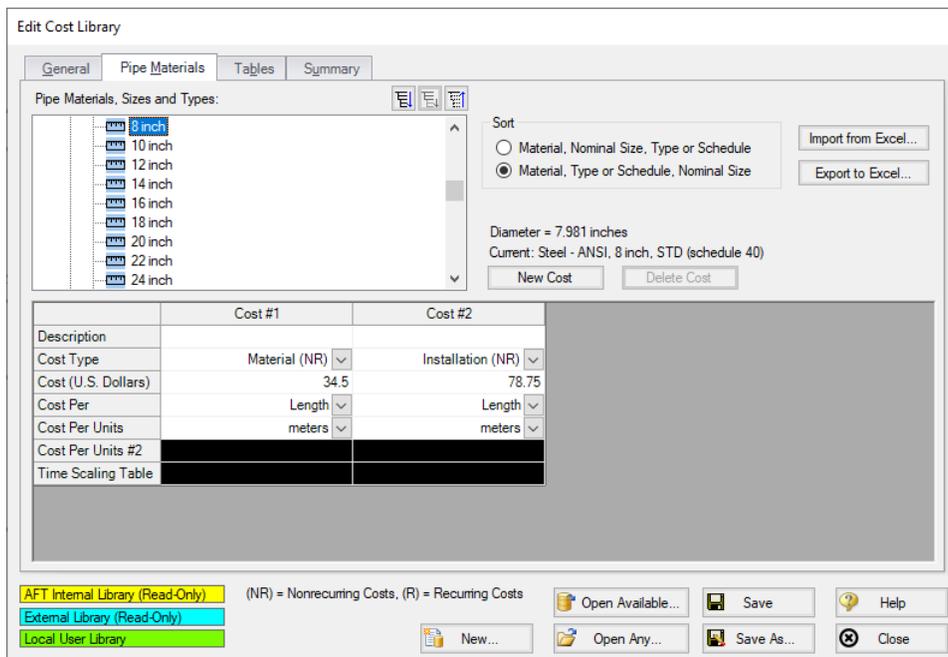


Figure 4: Non-recurring pipe costs, such as material and installation costs, are entered on the Pipe Materials tab

After the material and installation costs are entered for each STD type for the sizes listed above, click Save to save the information in the cost library file, and then click Close to close the Edit Cost Library window.

Create a cost library for the elbow fittings with cost scale tables

Several pipes in the model have 90 degree elbows specified in the Fittings and Losses tab in the respective Pipe Properties windows. The cost of these 90 degree elbow fittings can be included in the cost

calculation. The costs for these items will be accounted for in scale tables. Scale tables can be used to vary a cost with a parameter such as diameter (i.e., size). Once created, this scale table can be applied to fittings and losses items.

Create a cost library for the elbow fittings and losses costs by opening the Library menu and selecting Cost Library. Create a new cost library by clicking **New**. You are prompted to choose the engineering library with which the cost library will be associated.

➤ **The cost library for the elbow fittings and losses will be associated with the AFT INTERNAL LIBRARY**, as this is the engineering library in which the pipe fittings and losses data is kept for the default elbow pipe fittings and losses. Select AFT INTERNAL LIBRARY and click Select.

On the General tab, specify the following information:

1. Cost Type = Monetary
2. Monetary Unit = U.S. Dollars
3. Description = Plant Cooling - Elbow Costs
4. Notes = This library is for the elbows included as additional Fittings and Losses for the pipes

Click Save and save this cost library with a file name such as ***Plant Cooling - Elbow Costs.cst***

The first table to create is the scale table for the 90-deg. elbow installation costs. Select the Tables tab, and click New Table.

Enter the following data for the scale table on the New Scale Table window:

1. Name = 90 Elbows Installation
2. Table Type = Diameter
3. Table Format = Cost

After entering the data, click OK.

Now enter the cost data in the table using the values in the table below, as shown in Figure 5. Note that the diameter units are in inches because the Steel - ANSI pipe material library uses inches. However, if you prefer to use a different unit for diameter, such as centimeters, AFT Fathom will make the appropriate conversions. For this example, use inches.

Diameter (inches)	Installation Cost (US dollars)
7.981	185.50
12.25	298.00
17.376	446.00
23.25	595.00
29.00	744.00
33.00	843.00

You do not need to enter data in the scale tables for every diameter in the model. If a diameter falls between two data points in the table, AFT Fathom will use the points on either side to linearly interpolate for a value. If the cost function is non-linear, you may need to add additional data points to achieve a more accurate cost value.

Plant Cooling Cost Calculation

Create another scale table for the material costs using the following data:

1. Name = 90 Elbows Material
2. Table Type = Diameter
3. Table Format = Cost

Diameter (inches)	Material Cost (US dollars)
7.981	68.00
12.25	151.50
17.376	262.50
23.25	373.50
29.00	484.50
33.00	558.50

After the scale tables have been created, and the cost data entered, the Tables tab should appear as shown in Figure 5.

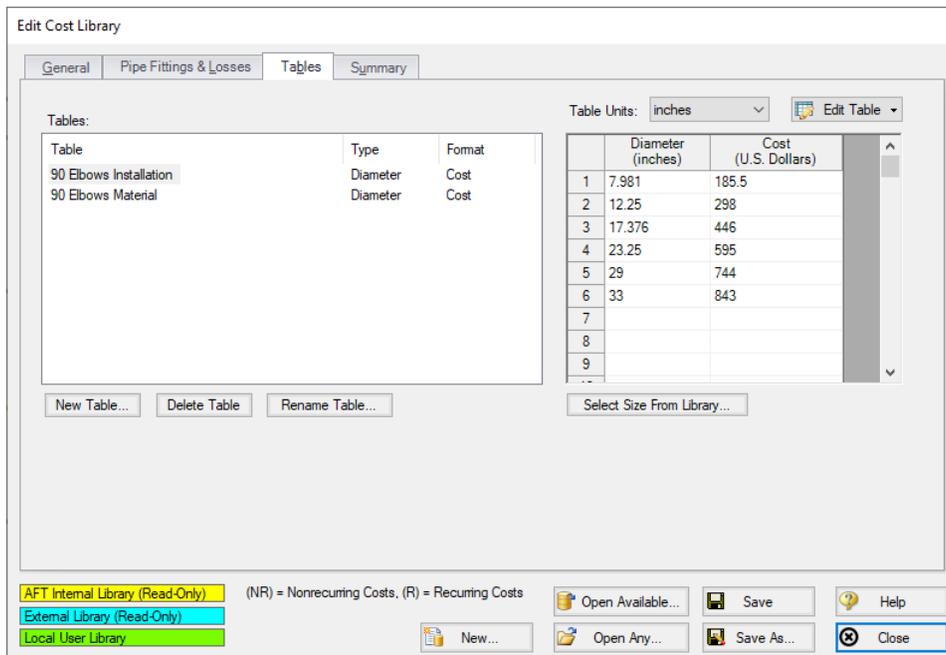


Figure 5: The Tables tab on the Edit Cost Library window is used to create cost scale tables

Click Save after the two cost tables are created and keep the Edit Cost Library window open for the next step.

Add the costs for pipe fittings

With the cost scale tables for the fittings defined, the costs for the pipe fittings can be added. Select the Pipe Fittings & Losses tab. A list of all of the available fittings and losses is displayed in the Pipe Fittings & Losses list.

The fittings in the model are Elbow/Bend, Smoothed Flanged, $r/D=1$, 90 deg. (C). Navigate through the list until you find the proper selection for these fittings.

Add the material cost by clicking New Cost, and entering the following data:

1. Description = Elbow Material Costs
2. Cost Type = Material (NR)
3. Material = Steel - ANSI
4. Material Size = All Sizes
5. Material Type = All Types
6. Use Size Table = Table of Costs
7. Multiplier = 1
8. Size Scaling Table = 90 Elbows Material

Now add the installation cost by clicking New Cost again and entering the following data:

1. Description = Elbow Installation Costs
2. Cost Type = Installation (NR)
3. Material = Steel - ANSI
4. Material Size = All Sizes
5. Material Type = All Types
6. Use Size Table = Table of Costs
7. Multiplier = 1
8. Size Scaling Table = 90 Elbows Installation

The actual cost values used for the fittings are the values that were entered in the scale tables on the Tables tab. The scale table to use for each cost is specified when the cost is defined. After the cost data is entered, the Pipe Fittings & Losses tab should appear as shown in Figure 6.

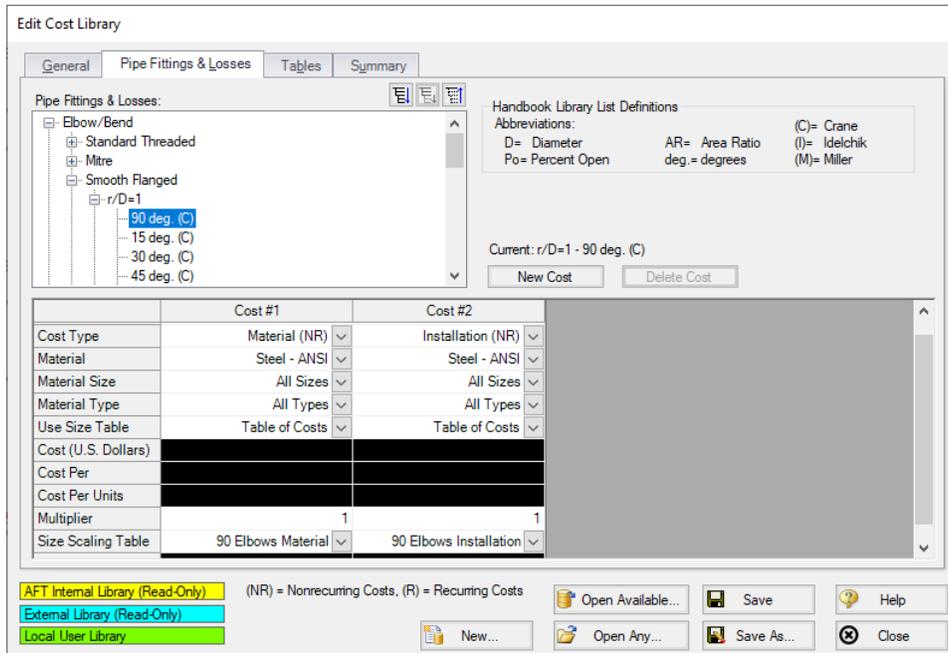


Figure 6: The Pipe Fittings & Losses tab in the Edit Cost Library window is used to specify costs for pipe fittings and losses items

Click Save and close the Edit Cost Library window, as now, the cost library for the elbow fittings and losses is now completed.

Create a cost library for the pumps

To enter cost for a pump, or any other junction type, the junction must first be added to the an engineering library in the Library Manager. Cost data is then entered in a cost library associated with that engineering library. The pumps used in this example have already been added to an engineering library, but no costs have been created for it yet. Create a new cost library associated with the Plant Cooling engineering library by opening the Library menu and selecting Cost Library.

Click **New**, then select the **Plant Cooling** library and click **Select**. Specify the following general information on the General tab:

1. Cost Type = Monetary
2. Monetary Unit = U.S. Dollars
3. Description = Plant Cooling - Pump Costs
4. Notes = This library is for the pumps in the Plant Cooling engineering library

Save the cost library to a file by clicking Save and giving the file a name such as **Plant Cooling - Pump Costs.cst**. When prompted, choose to add the new library to the list of available and connected libraries. Keep the Edit Cost Library window open for the next step.

Enter the pump costs

After the pump cost library has been created, select the Junctions tab. This is where you will enter the material and installation costs for the pumps.

All of the junctions that are available in the selected engineering library will be listed. Select the **Metric - Plant Cooling Pump** from the list of pumps.

Add the material cost by clicking New Cost, and entering the following data:

1. Description = Pump Material Cost
2. Cost Type = Material (NR)
3. Use Size Table = (None)
4. Cost (U.S. Dollars) = 28500
5. Cost Per = Item

Add the installation cost by clicking New Cost again and entering the following data:

1. Description = Pump Installation Cost
2. Cost Type = Installation (NR)
3. Use Size Table = (None)
4. Cost (U.S. Dollars) = 12500
5. Cost Per = Item

Figure 7 shows the pump costs entered in the cost library.

Click Save and then click Close.

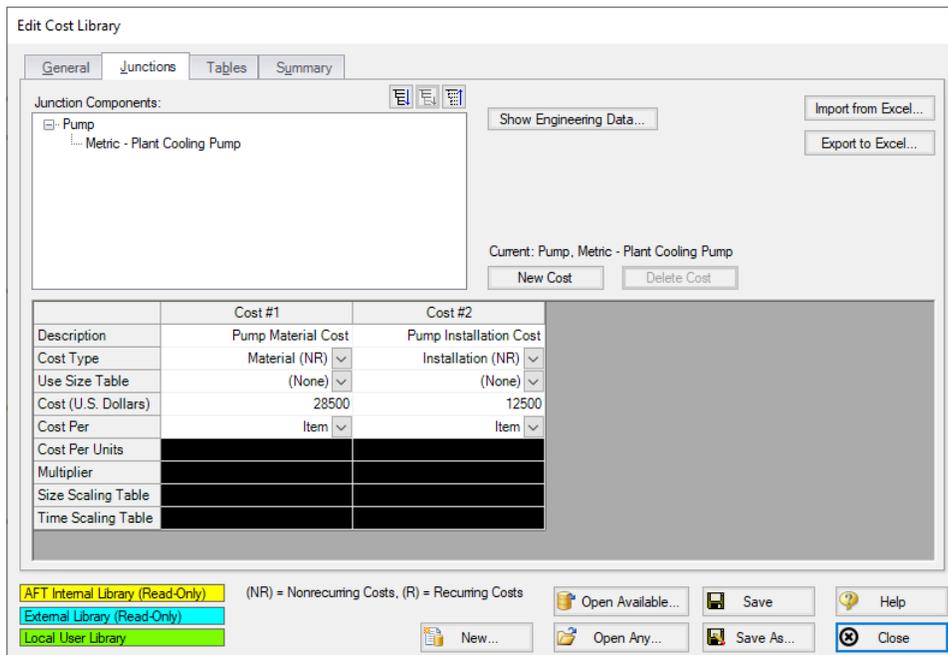


Figure 7: The Junctions tab on the Edit Cost Library window is used to specify costs for junctions

Connecting the cost libraries

The Plant Cooling engineering and cost libraries should have been added to the list of available libraries and been connected to the example model. If they were not, you can use the Library Manager to connect the cost libraries now (see Figure 9). The **Plant Cooling - Pump Costs** is connected subordinate to the **Plant Cooling**. The **Plant Cooling - Pipe Costs Metric** is connected subordinate to the **Steel - ANSI** pipe material library. Lastly, the **Plant Cooling - Elbow Costs** is connected subordinate to the **AFT INTERNAL LIBRARY**. Refer to the AFT Fathom Help Content for information on how to connect libraries using the Library Manager.

Make sure that the pumps are utilizing the engineering library junction data, otherwise the costs will not be calculated.

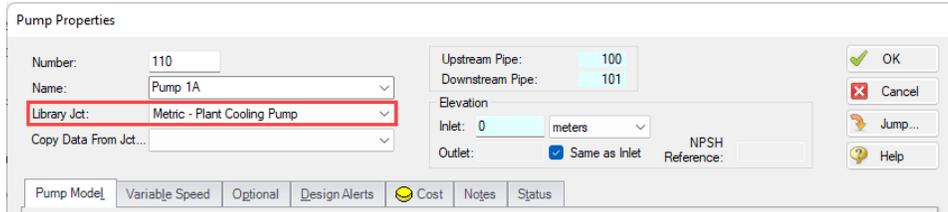


Figure 8: Pump using the engineering library data

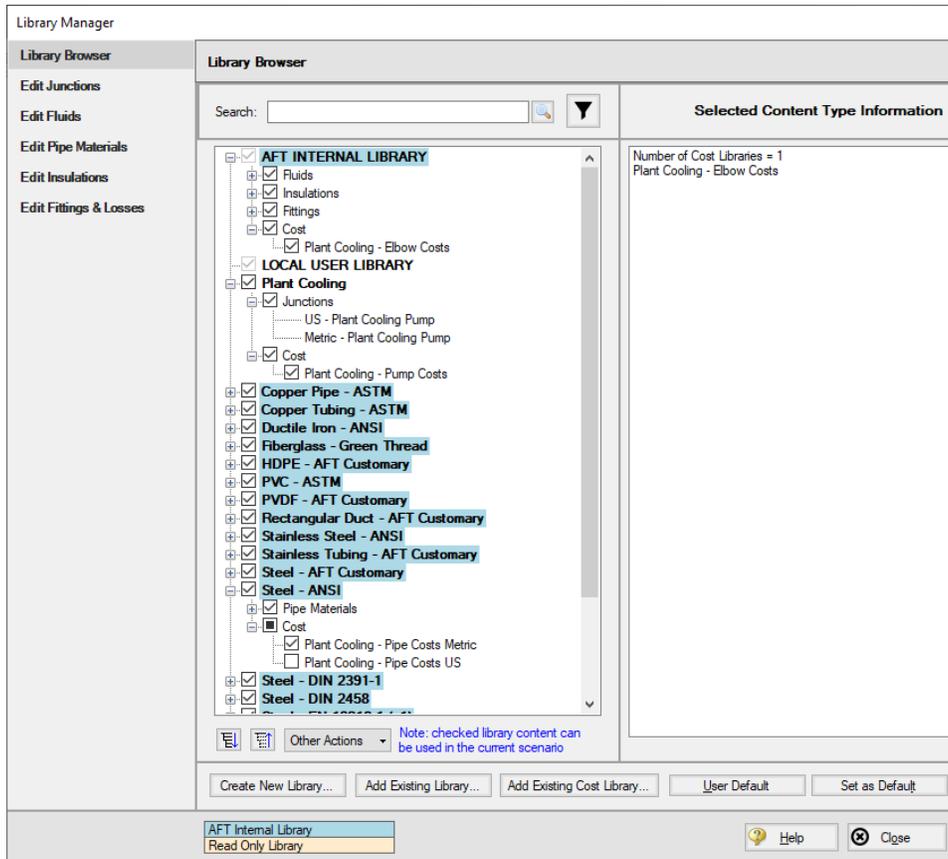


Figure 9: Library Manager with connected engineering and cost libraries

Step 5. Including objects in the Cost Report

The final step before performing a cost analysis is to specify which pipes and junctions you want to be included in the final cost report. This is done by opening the properties window, navigating to the Cost tab, and choosing Include in Cost Report for each pipe and junction to be included, as shown for Pipe 100 in Figure 10.

Include the four circulating water pumps and all of the pipes in the cost report. The Global Edit feature may be used to update this information for all of the pumps and pipes.

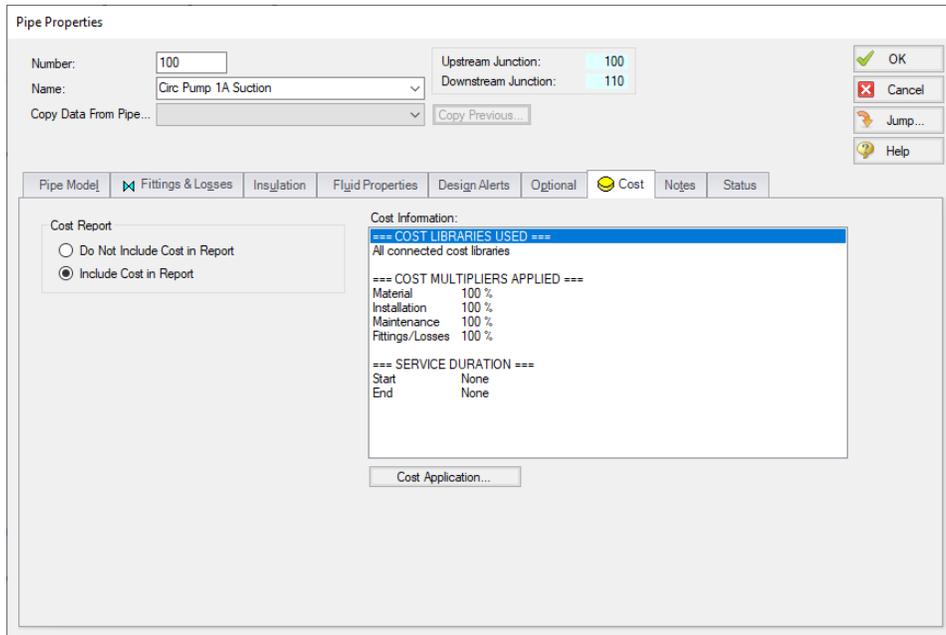


Figure 10: The Cost tab on the Pipe and Junction Properties windows is used to include the objects in the Cost Report

Step 6. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 7. Examine the Cost Report

The Cost Report is displayed in the General Results section of the Output window. View the Cost report by selecting the Cost Report tab. The content of the Cost Report can be modified from the Output Control window. Figure 11 shows the Cost Report for this example with the Material, Installation, Operation/Energy, and Total costs displayed.

The Cost Report shows the total system cost, as well as the individual totals for the material, installation, and energy costs. In addition, the Cost Report displays the detailed cost for each pipe, junction, and fitting that was included in the report. The items are grouped together by type, and a subtotal for each category is listed.

Plant Cooling Cost Calculation

General Warnings Design Alerts Cost Report Pump Summary Valve Summary Heat Exchanger Summary Reservoir Summary										
Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL	
TOTAL OF ALL MODEL COSTS										4,589,100
Total of All Shown Costs			413,515	687,646	1,101,160	0	3,487,938	3,487,938	4,589,099	
Pipe Subtotal			287,112	617,622	904,734	0	0	0	904,734	
P100	Pipe	Circ Pump 1A Suction	349	842	1,190	0	0	0	1,190	
P101	Pipe	Circ Pump 1A Disch	698	1,683	2,381	0	0	0	2,381	
P102	Pipe	Circ Pump 1B Suction	349	842	1,190	0	0	0	1,190	
P103	Pipe	Circ Pump 1B Disch	698	1,683	2,381	0	0	0	2,381	
P104	Pipe	Supply 1	59,100	126,975	186,075	0	0	0	186,075	
P105	Pipe	Cond 1 In	2,855	5,415	8,270	0	0	0	8,270	
P106	Pipe	Cond 1 Out	2,855	5,415	8,270	0	0	0	8,270	
P107	Pipe	LO Clr 1 In	207	473	680	0	0	0	680	
P108	Pipe	LO Clr 1 Out	207	473	680	0	0	0	680	
P109	Pipe	Return 1	59,100	126,975	186,075	0	0	0	186,075	
P110	Pipe	CT 1 Hdr A	1,575	3,801	5,376	0	0	0	5,376	
P111	Pipe	CT 1 Hdr B	1,080	2,757	3,837	0	0	0	3,837	
P112	Pipe	Riser 1/1	888	2,028	2,916	0	0	0	2,916	
P113	Pipe	Riser 1/2	888	2,028	2,916	0	0	0	2,916	
P114	Pipe	Riser 1/3	888	2,028	2,916	0	0	0	2,916	
P200	Pipe	Circ Pump 2A Suction	349	842	1,190	0	0	0	1,190	
P201	Pipe	Circ Pump 2A Disch	698	1,683	2,381	0	0	0	2,381	
P202	Pipe	Circ Pump 2B Suction	349	842	1,190	0	0	0	1,190	
P203	Pipe	Circ Pump 2B Disch	698	1,683	2,381	0	0	0	2,381	
P204	Pipe	Supply 2	59,100	126,975	186,075	0	0	0	186,075	
P205	Pipe	Cond 2 In	2,855	5,415	8,270	0	0	0	8,270	

Figure 11: The Cost Report in the General Output section shows the results of the cost analysis

Analysis Summary

The cost analysis for this example shows the following costs for the plant cooling system design:

Total of All Model Costs	\$4,589,100
Total Material Cost	\$413,515
Total Installation Cost	\$687,646
Total Operation/Energy Cost	\$3,487,938
Total Cost of Pipe	\$904,734
Total Cost of Pumps	\$3,651,938
Total Cost of Fittings	\$32,426

Minimizing Cost with AFT Fathom Automated Network Sizing Module™

The AFT Fathom [Automated Network Sizing Module \(ANS\)](#) is a tool built on AFT Fathom technology. With the ANS module you can automatically size all pipes or ducts in your system to minimize monetary cost, weight, volume, or surface area. In addition, you can concurrently size the pumps and pipes to obtain the absolute lowest cost system that satisfies your design requirements. Finally, by accounting for non-recurring and recurring costs, you can size pipe and duct systems to minimize life cycle costs over some specified duration.

Plant Cooling Cost Calculation

By using the AFT Fathom Automated Network Sizing Module (ANS) on this example, not only could the system cost be determined, the cost could also be minimized by varying selected pipe and junction sizes in the system.

Non-Newtonian Mill Discharge Slurry

Summary

This example demonstrates how one can take small-scale raw rheological data for a mill slurry at a minerals plant and use it on a full-scale pipe calculation.

Topics Covered

- Reviewing and entering rheological data
- Using homogeneous scale-up features
- Obtaining and using Power Law and Bingham Plastic constants

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

In addition the user should have worked through [Non-Newtonian Phosphates Pumping](#) example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Non-Newtonian Mill Discharge Slurry.fth***
- ***Non-Newtonian Mill Discharge Slurry Data.xlsx***

Problem Statement

The piping for a non-settling slurry for a mill discharge at a minerals plant is being designed. The system will supply 200 m³/hr of slurry through a 12-inch (305 mm) horizontal pipe 100 meters long.

Rheological data using the actual fluid in a 1/4 inch (1 cm) pipe has been obtained. Perform non-settling slurry calculations to evaluate how well the Bingham Plastic and Power Law models follow the raw data. Then determine the pressure drop in the full-scale pipe using Bingham Plastic, Power Law and Homogeneous Scale-up models.

Step 1. Review Raw Data in Excel

The data taken for 1/4 inch STD schedule (0.9246 cm ID), 0.75 meter pipe is as follows:

Density = 1400 kg/m³

Table 1: Pressure drop data on mill slurry (1/4 inch pipe - 0.9246 cm ID, 0.75 meter long)

Run	Q (m ³ /hr)	dP (Pa)
1	0.003	25,000
2	0.015	36,000
3	0.025	44,000
4	0.050	53,000
5	0.100	60,000

The Bingham Plastic and Power Law models typically process data in terms of **shear rate** and **shear stress**. The above data needs to be changed into a form for input into AFT Fathom.

Open the provided Excel spreadsheet **Non-Newtonian Mill Discharge Slurry Data.xlsx**. Click the Excel tab for Metric units. Here one can see the raw data and calculated shear rate and shear stress. See Figure 1 for a view of the data. The shear rate and shear stress data is in the yellow columns.

Tube Diameter	0.009246 meters	
Tube Length	0.75 meters	
Slurry Density	1400 kg/m ³	
Gravity	9.81 meters/sec ²	

Run #	Test Data		Calculated Data			Data Input to Fathom		Fathom Results		
	Flow rate m ³ /hr	ΔP Pascals	Flow rate m ³ /sec	Velocity meters/sec	Jm m/m	Shear Rate 8v/D 1/seconds	Shear Stress (Tau) PD/4L Pascals	Vol. Flow m ³ /sec	Bingham Plastic dP Stag. Pascals	Power Law dP Stag. Pascals
1	0.003	25,000	8.333E-07	0.0124	2.43	10.7	77.0	8.33E-07	34,082	24,830
2	0.015	36,000	4.167E-06	0.0621	3.49	53.7	110.9	4.17E-06	38,869	37,592
3	0.025	44,000	6.944E-06	0.1034	4.27	89.5	135.6	6.94E-06	41,794	42,881
4	0.050	53,000	1.389E-05	0.2069	5.15	179.0	163.3	1.39E-05	47,996	51,267
5	0.100	60,000	2.778E-05	0.4137	5.82	358.0	184.9	2.78E-05	58,840	61,294

Figure 1: Mill slurry raw and calculated data from Excel file

Step 2. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel:
 - a. Fluid Library = User Specified Fluid
 - b. Name = Mill Discharge Slurry
 - c. Density = 1400 kg/m³
 - d. Dynamic Viscosity = 1 Pa-sec (note that the viscosity entered here will not be used once the non-Newtonian parameters are entered)
3. Open the Viscosity Model panel
 - a. Viscosity Model = Bingham Plastic
 - b. Select Calculate from Rheological Data
 - c. Click Calculate Constants
 - d. Raw Data Type = Tube Flow Rheometer Data (8V/D)
 - e. Raw Data = (make sure to change the units for Shear Stress to Pascals)

Parameter	Shear Rate	Shear Stress
Units	1/seconds	Pascals
1	10.7	77.0
2	53.7	110.9
3	89.5	135.6
4	179.0	163.3
5	358.0	184.9

-
-
-
-
-
- f. Curve Fit Order = 1

Calculate Bingham Plastic Constants

Rheological Data

Raw Data Type: Tube Flow Rheometer Data (8V/D) Shear Rheometer Data (du/dy) ?

Parameter	Shear Rate 8V/D	Shear Stress Tau
Units	1/seconds	Pascals
1	10.7	77
2	53.7	110.9
3	89.5	135.6
4	179	163.3
5	358	184.9
6		
7		
8		
9		
10		
11		
12		
13		
14		

Curve Fitting
Curve Fit Type: Polynomial

Shear Stress

Curve Fit Order: 1

Generate Curve Fit Now

Edit Table

OK Cancel Help

Figure 2: Bingham Plastic data entry

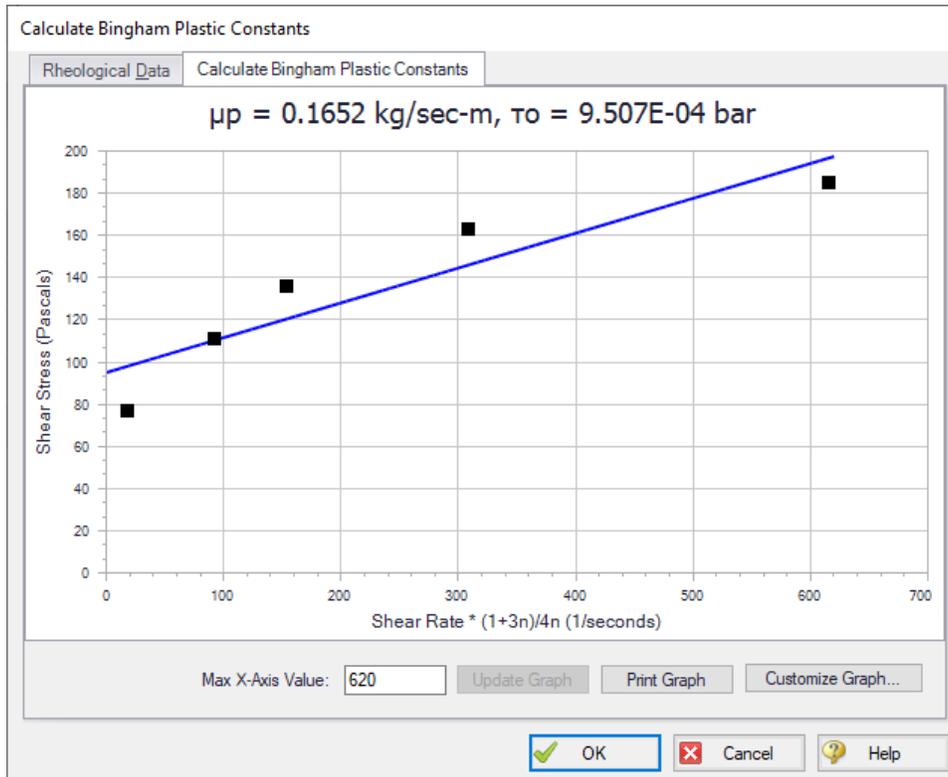


Figure 3: Bingham Plastic curve fit and constants calculation

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

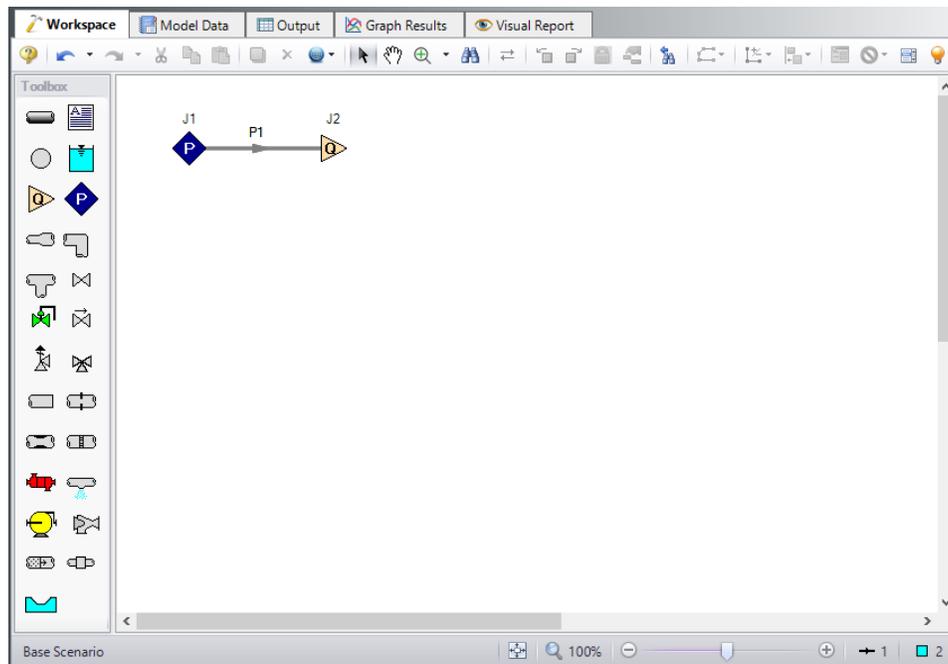


Figure 4: Layout of pipe system for Mill Discharge Slurry example

Pipe Properties

1. Pipe Material = Steel - ANSI
2. Pipe Geometry = Cylindrical Pipe
3. Size = 1/4 inch
4. Type = STD (schedule 40)
5. Length = 0.75 meters

Junction Properties

1. J1 Assigned Pressure
 - a. Elevation = 0 meters
 - b. Pressure = 10 bar (1000 kPa)
 - c. Pressure Specification = Stagnation
2. J2 Assigned Flow
 - a. Elevation = 0 meters
 - b. Type = Outflow
 - c. Flow Specification = Volumetric Flow Rate
 - d. Flow Rate = 0.003 m³/hr

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will

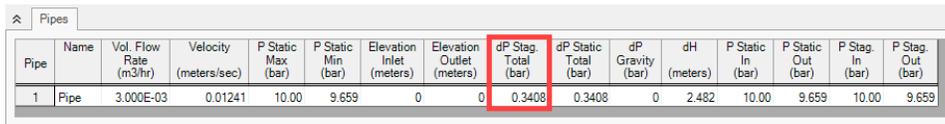
have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

Figure 5 shows the Output window. It can be seen that the pressure drop is 0.3408 bar (34,082 Pascals) for 0.003 m³/hr, compared to the raw data value of 0.2500 bar (25,000 Pascals).



Pipe	Name	Vol. Flow Rate (m ³ /hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)
1	Pipe	3.000E-03	0.01241	10.00	9.659	0	0	0.3408	0.3408	0	2.482	10.00	9.659	10.00	9.659

Figure 5: The Output window with default output data

Step 7. Create Scenarios

From the Base Scenario, make a child scenario named *Bingham Plastic*.

Step 8. Run Other Bingham Plastic Scenarios

Load the Bingham Plastic scenario and use the Duplicate Special feature to create five identical runs. Enter flow rates of 0.015, 0.025, 0.050 and 0.100 m³/hr at the Assigned Flow junctions. Run the model and note the pressure drop for each pipe.

Step 9. Create and Run Power Law Scenarios

In the Scenario Manager, right click the Bingham Plastic scenario and select **Clone Without Children**. Name this scenario *Power Law*.

1. Load the Power Law Scenario
2. Open Analysis Setup
3. Open the Viscosity Model panel

- a. Viscosity Model = Power Law
 - b. Select Calculate from Rheological Data
 - c. Click Calculate Constants
 - d. Raw Data Type = Tube Flow Rheometer Data (8V/D)
 - e. Raw Data = should still be loaded from before
 - f. Curve Fit Order = 1
 - g. Click Generate Curve Fit Now
 - h. Click OK
4. Click OK

Run the model again.

Figure 7 shows the output for each of the ten cases. Figure 8 plots the test data against the results of Bingham Plastic and Power Law. From Figure 8 it is clear the Power Law model fits the mill discharge slurry data better than Bingham Plastic.

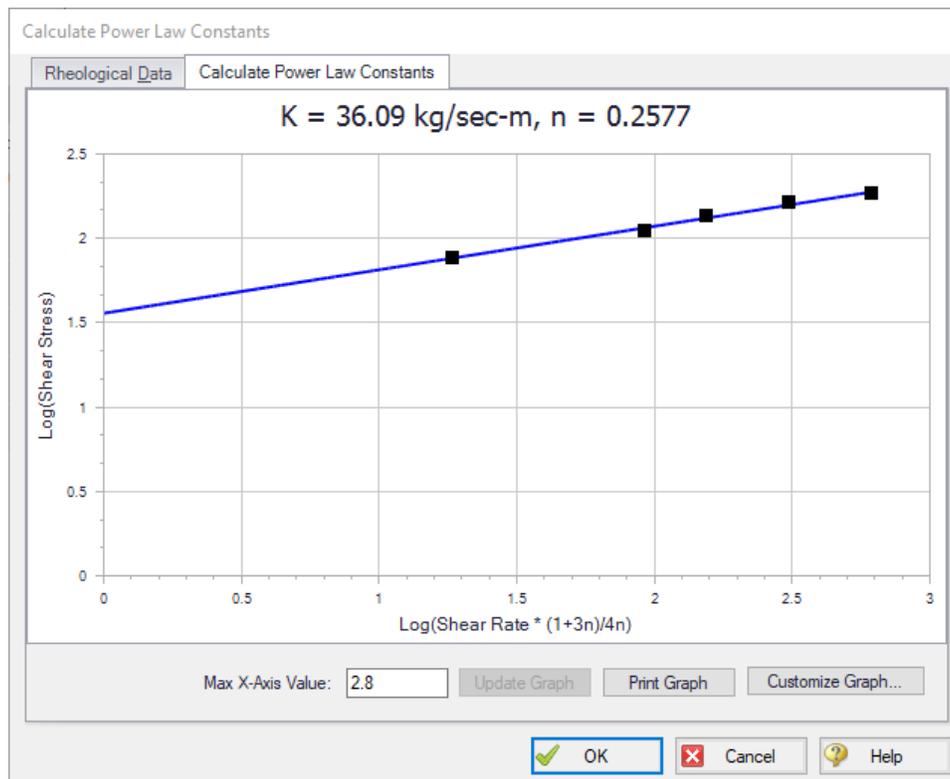


Figure 6: Power Law curve fit from Viscosity Model panel

Non-Newtonian Mill Discharge Slurry

Tube Diameter 0.009246 meters
 Tube Length 0.75 meters
 Slurry Density 1400 kg/m³
 Gravity 9.81 meters/sec²

Run #	Test Data		Calculated Data			Data Input to Fathom		Fathom Results		
	Flow rate m ³ /hr	ΔP Pascals	Flow rate m ³ /sec	Velocity meters/sec	Jm m/m	Shear Rate $\delta v/D$ 1/seconds	Shear Stress (Tau) PD/4L Pascals	Vol. Flow m ³ /sec	Bingham Plastic dP Stag. Pascals	Power Law dP Stag. Pascals
1	0.003	25,000	8.333E-07	0.0124	2.43	10.7	77.0	8.33E-07	34,082	24,830
2	0.015	36,000	4.167E-06	0.0621	3.49	53.7	110.9	4.17E-06	38,869	37,592
3	0.025	44,000	6.944E-06	0.1034	4.27	89.5	135.6	6.94E-06	41,794	42,881
4	0.050	53,000	1.389E-05	0.2069	5.15	179.0	163.3	1.39E-05	47,996	51,267
5	0.100	60,000	2.778E-05	0.4137	5.82	358.0	184.9	2.78E-05	58,840	61,294

Figure 7: Summary of AFT Fathom output and test data

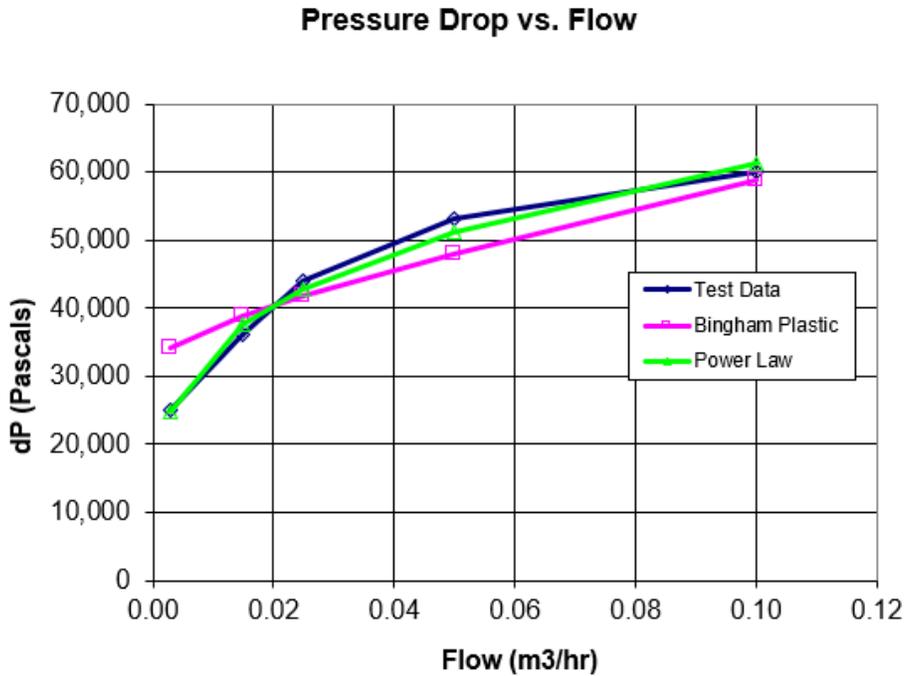


Figure 8: Graph comparing AFT Fathom output and test data

Step 10. Evaluate Full Scale Data

1. Make another child scenario called Full Scale - 12 inch
2. Change pipe P1 to 100 meters length and 12 inch diameter
3. Change junction J2 to a flow rate of 200 m³/hr
4. Make three child scenarios below Full Scale - 12 inch: one for each, **Bingham Plastic**, **Power Law**, and **Homogeneous Scale-up**

5. Load the Bingham Plastic scenario, open Analysis Setup, and perform a curve fit for Bingham Plastic
6. Run the Bingham Plastic scenario and note the pressure drop
7. Repeat the above for the Power Law scenario
8. For the Homogeneous Scale-up scenario, use the test data from the Excel spreadsheet. Make sure to enter the data as Volumetric in m³/hr and Jm in m/m. All 5 data points are laminar flow, as shown in Figure 9.

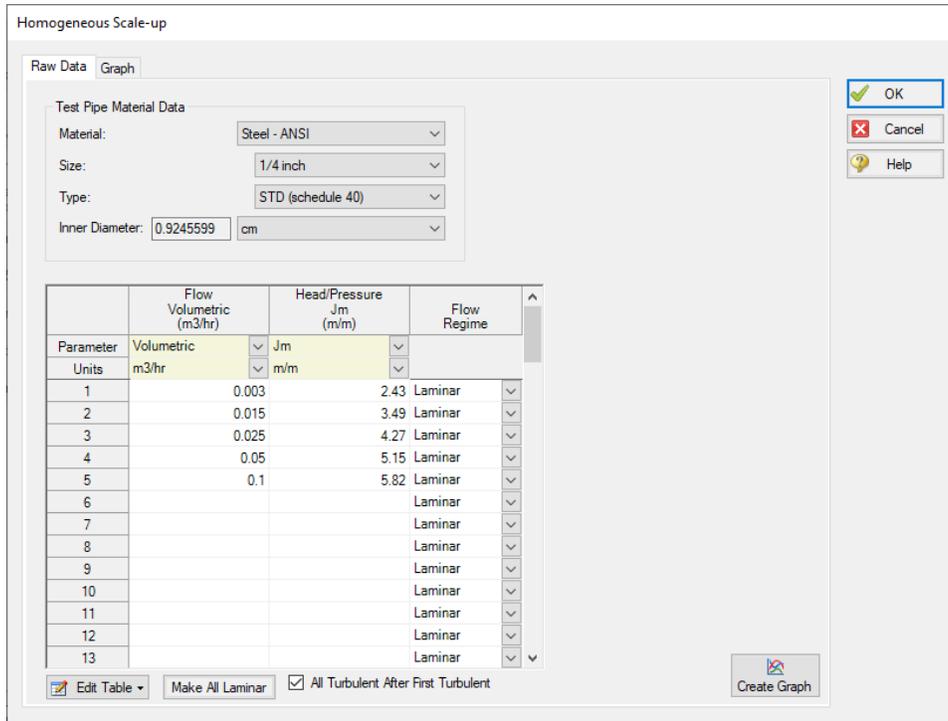


Figure 9: Homogeneous Scale-up window with test data input

Figure 10 shows a comparison of pressure drop results using multi-scenario output for the three children. It can be seen that there is closer agreement between Power Law and Homogeneous Scale-up. The Bingham Plastic results are not as close which was true earlier in this example comparing to actual test data (Figure 8). Therefore the Bingham Plastic pressure drop should be excluded and the design pressure drop should be based on Power Law and Homogeneous Scale-up.

Non-Newtonian Mill Discharge Slurry

Pipes															
Pipe	Name	Vol. Flow Rate (m ³ /hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag In (bar)	P Stag Out (bar)
1 - Bingham Plastic	N/A	200.0	0.7614	9.996	8.564	0	0	1.432	1.432	0	10.427	9.996	8.564	10.00	8.568
1 - Power Law	N/A	200.0	0.7614	9.996	8.817	0	0	1.178	1.178	0	8.584	9.996	8.817	10.00	8.822
1 - Homogeneous Scale-up	Pipe	200.0	0.7614	9.996	8.889	0	0	1.107	1.107	0	8.063	9.996	8.889	10.00	8.893

All Junctions									
Assigned Flow		Assigned Pressure							
Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag In (bar)	P Stag Out (bar)	Vol. Flow Rate Thru Jct (m ³ /hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)	
1 - Bingham Plastic	N/A	9.996	9.996	10.000	10.000	200.0	77.78	0	
1 - Power Law	N/A	9.996	9.996	10.000	10.000	200.0	77.78	0	
1 - Homogeneous Scale-up	Assigned Pressure	9.996	9.996	10.000	10.000	200.0	77.78	0	
2 - Bingham Plastic	N/A	8.564	8.564	8.568	8.568	200.0	77.78	0	
2 - Power Law	N/A	8.817	8.817	8.822	8.822	200.0	77.78	0	
2 - Homogeneous Scale-up	Assigned Flow	8.889	8.889	8.893	8.893	200.0	77.78	0	

Figure 10: Full scale pressure drop comparison of Bingham Plastic, Power Law, and Homogeneous Scale-up

Analysis Summary

A non-settling mill discharge slurry for a minerals plant was modeled using non-settling slurry calculations in AFT Fathom. Runs for small scale and full-scale calculation were made and pressure drop was determined for these cases.

Flow Induced Vibration Calculations (Metric Only)

Summary

Repeated vibration on a piping system can result in vibration induced fatigue and failure. Pipe failure due to vibration creates safety concerns for personnel and environmental contamination, as well as costly downtime and repairs. Despite these concerns, vibration sources and resulting fatigue are often not addressed by design standards. Instead, designers and operators often react to vibration issues rather than prevent and avoid these issues proactively.

In response to vibration concerns, the [Energy Institute](#) developed the [Guidelines for the Avoidance of Vibration Induced Fatigue Failure in Process Pipework](#). These guidelines provide sources of vibration for engineers to consider, many of which can be identified or modeled in an AFT application as found in the Table 1 below.

Table 1: Vibration sources and applicable AFT Application

Vibration Sources		AFT Application
Surge	Sudden changes to flow/- pressure	AFT Impulse / AFT xStream
Cavitation	Vapor formation and collapse	AFT Fathom / AFT Impulse
Reciprocating Equipment	Identify resonant frequencies to avoid during operation	AFT Impulse PFA Module / AFT xStream PFA Module
FIV	Turbulence from flow discontinuities	AFT Fathom / AFT Arrow
AIV	High frequency acoustic energy	AFT Arrow
FIP	Excitation at dead end branches	AFT Arrow

These guidelines provide a structured approach to identify, assess, and correct vibration concerns through qualitative and quantitative methods. AFT has developed a Flow Induced Vibration Calculations spreadsheet which, in conjunction with model output and user input, can perform key parts of the recommended quantitative assessment of different vibration sources. These calculations used with the Energy Institute guidelines will help engineers identify vibration concerns in their systems.

In this example, learn how to use AFT Fathom to help calculate the Likelihood of Failure (LOF) for **Flow Induced Vibration (FIV)** in accordance with Energy Institute Guidelines to avoid vibration induced fatigue failure.

Topics Covered

- The cause and concerns of **Flow Induced Vibration (FIV)**:
 - Occurs in both compressible and incompressible flow
 - Excitation caused by turbulence from discontinuities or boundary layer separation
 - Generally concentrated at low frequencies (<100 Hz)
 - Can lead to visible vibrations of piping and supports
- Required calculation parameters for FIV
- Excel Export Manager
- Use of AFT vibration analysis spreadsheet

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Flow Induced Vibration Calculations.fth***
- ***Flow Induced Vibration Calculations.xlsx***

Problem Statement

Turbulence is common in most practical piping systems. In this example, we will examine turbulence in straight pipes, caused by the boundary layer at the pipe wall, with its intensity depending on the flow regime, characterized by the Reynolds number. However, in typical scenarios, major flow disturbances like process equipment, partially closed valves, sharp bends, tees, and reducers are the primary sources of turbulence.

Reference: Pages 48-53, Section T2.2, Flowchart T2-1

Purpose: Looking at the part of the mainline with most turbulence and the greatest distance between supports to determine the LOF

Input Values/Sources/Descriptions:

Note: Table 2 shows the specific values used in this model, but the steps in this Example can be used to calculate the LOF for vibration concerns in any model.

Table 2: Input Values/Sources/Descriptions

Object	Inputs	Example Value	Source	Description
P4 (Pipe)	L,span	7 m	User Input	Max span length between supports
P4 (Pipe)	T	9.525 mm	User Input	Thickness of pipe wall
P4 (Pipe)	fn (adv. only)	1 Hz	User Input	Structural natural frequencies
P4 (Pipe)	D,int	438.15 mm	AFT Export	Internal pipe diameter
P4 (Pipe)	ρ	998.9342 kg/m ³	AFT Export	Fluid density (static)
P4 (Pipe)	v	4.0503 m/s	AFT Export	Fluid velocity

Note: L,span of 7 m assumes that supports are at the elbows and the pressure boundaries for the example model.

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the Model

Open the **Metric - Flow Induced Vibration Calculations.fth** model file.

The workspace should look like Figure 1 below.

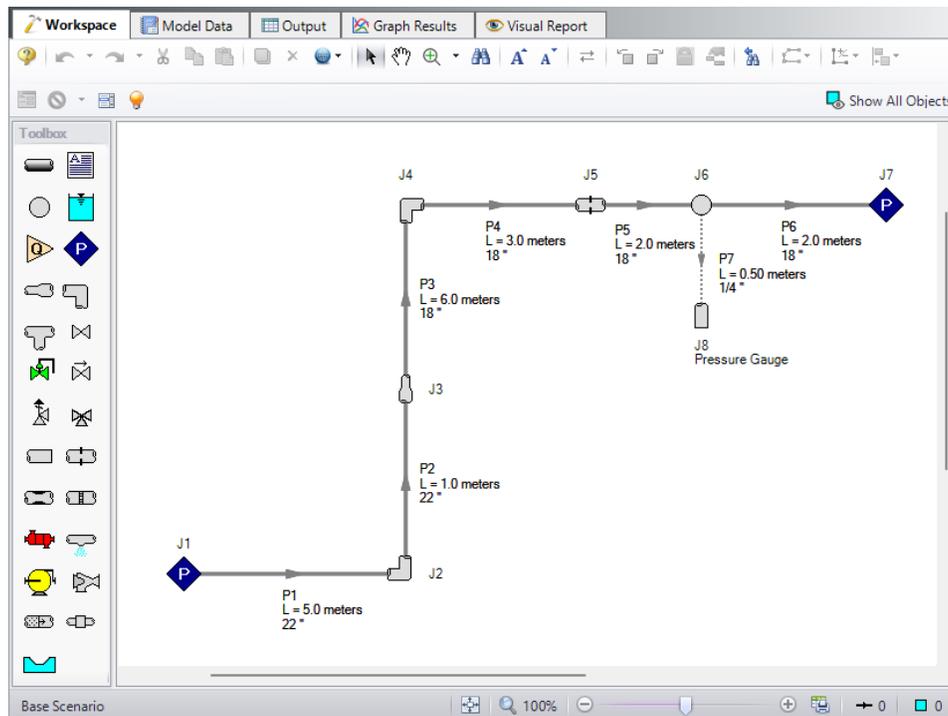


Figure 1: Layout of FIV Analysis Model on the Workspace

Step 3. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 4. Excel Export Manager

1. From the Workspace, click the **Excel Export Manager** button on the toolbar, then click **Open**.

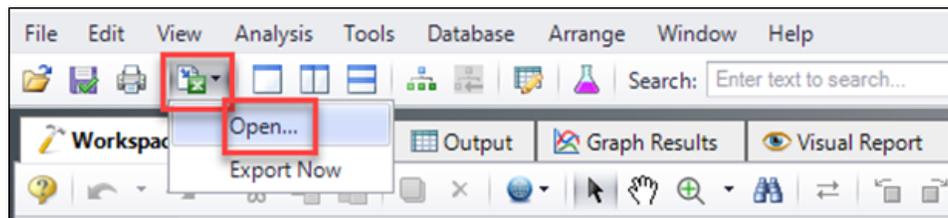


Figure 2: Opening Excel Export Manager from the Workspace

2. The [Excel Export Manager](#) window appears.

Flow Induced Vibration Calculations

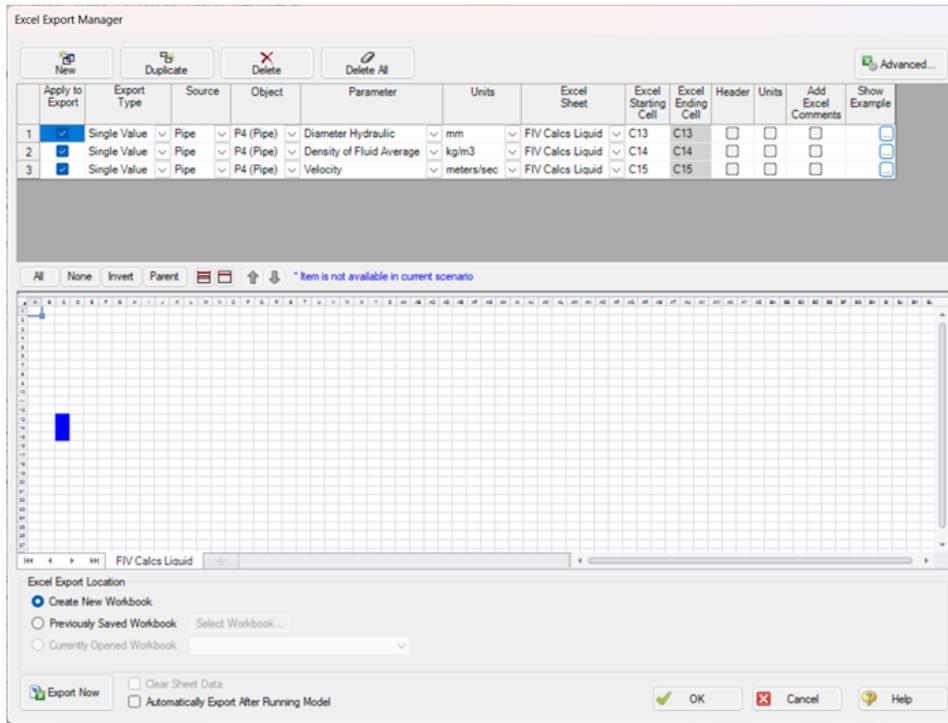


Figure 3: Excel Export Manager window

Note that three rows have been added to the Excel Export Manager with the following information. P# (Pipe) is the same pipe for all rows.

Table 3: Excel Export Manager parameters for FIV calculations

Object	Parameter	Units	Excel Sheet	Excel Starting Cell	Header	Units
P# (Pipe)	Diameter Hydraulic	mm	FIV Calcs Liquid	C13	<input type="checkbox"/>	<input type="checkbox"/>
P# (Pipe)	Density of Fluid Average	kg/m3	FIV Calcs Liquid	C14	<input type="checkbox"/>	<input type="checkbox"/>
P# (Pipe)	Velocity	meters/sec	FIV Calcs Liquid	C15	<input type="checkbox"/>	<input type="checkbox"/>

3. At the bottom, for Excel Export Location, click the radio button for **Previously Saved Workbook**.
4. The Flow Induced Vibration Calculation Excel spreadsheet should be preselected. If not, click the **Select Workbook** button, navigate to the **Flow Induced Vibration Calculation.xlsx** file then click **Open**. The top of the Excel Export Manager window should look like Figure 4 and the bottom should look like Figure 5.

Flow Induced Vibration Calculations

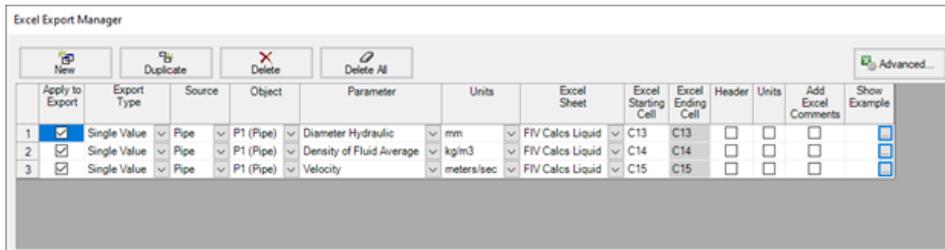


Figure 4: Excel Export Manager window with incompressible fluids

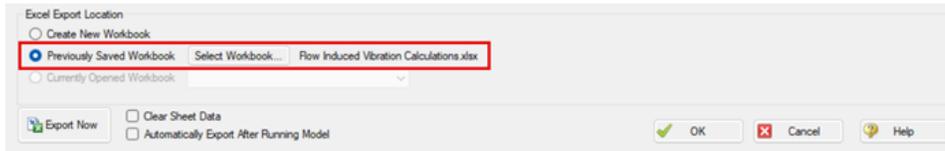


Figure 5: Bottom of the Excel Export Manager window

- Once all inputs are confirmed to match the above window, click **Export Now** at the bottom left.



Figure 6: Bottom of the Excel Export Manager window

Step 5. Vibration Analysis Spreadsheet

AFT Fathom will export the parameters to the spreadsheet which has the prebuilt calculations. To complete the vibration analysis, use the following instructions:

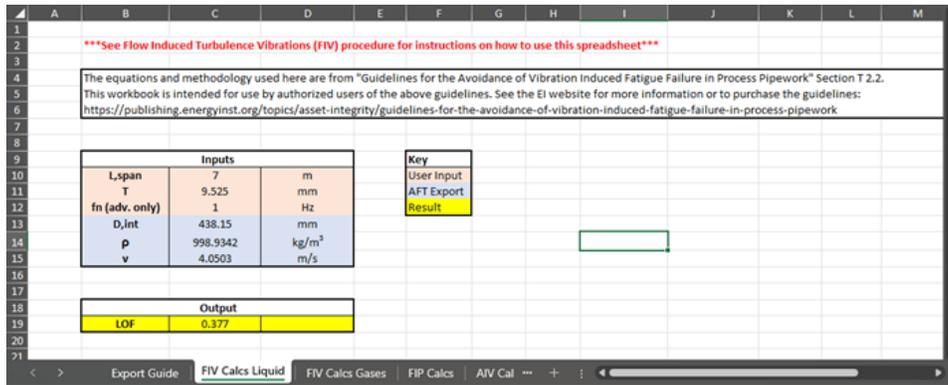


Figure 7: Flow Induced Vibration Calculations Excel document following export

1. Navigate to the **FIV Calcs Liquid** sheet in the Excel workbook.
2. With the values from AFT Fathom, note the user specified values for the remaining inputs:
 - a. L,span in m
 - b. T in mm
 - c. fn (advanced only)
 - For the highest accuracy, the natural frequency should be measured for the system.
 - If the natural frequency is not known the typical fundamental natural frequency for the system based on the support type is given in Table T2-1 on page 50 of the EI Guidelines and is shown in the spreadsheet in the “Determining Support Type” table in cells F28-F31.
3. The LOF is calculated and shown in cell C19 in the output section.
4. Use the EI Guidelines to determine appropriate actions based on displayed LOF.

Conclusion

An LOF of 0.377 indicates a moderate risk that the pipe may fail, but does not necessarily require a full redesign of the main line. When failure is not imminent (LOF < 0.5), the following is recommended by the EI Guidelines:

- Small bore connections on the main line should be assessed.
- A visual survey should be undertaken to check for poor construction and/or potential vibration transmission from other sources.

Module Walk-Through Examples

The examples in the Fathom Module Help System are designed to get new users trained to use the Fathom modules correctly in the shortest amount of time.

If this is your first time using Fathom, please start with the [Walk-Through Examples](#) to learn how to create and run Fathom models. Start with the [Three Reservoir Model](#), because it provides the most complete, basic description on how to set up and run a Fathom model. After this example, it is recommended that you work through the remaining Walk-Through examples.

If you are just learning to work with the Fathom modules, start with the Beginner model for each module, then choose an example from the tables on the Module pages that most closely matches your application. The examples are arranged in order of increasing complexity.

[Fathom GSC Examples](#)

[Fathom XTS Examples](#)

[Fathom SSL Examples](#)

[Fathom ANS Examples](#)

[Fathom Combined Module Examples](#)

Goal Seek and Control (GSC) Examples

The examples below utilize the Goal Seek and Control module.

Fathom GSC Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
Beginner - Heat Transfer in a Pipe	Beginner	Water (liquid)	1	0	Learn the basics of the GSC module. See how to use Analysis Setup to activate and enable the GSC module and to use the Goal Seek and Control group to define variables and goals. Learn where to find the GSC Variables and GSC Goals tabs in the Output window.
Pump Sizing and Selection with Flow Control Valves	Beginner	Water (liquid)	9	1	Learn how to use the GSC module to size a pump in a system with a pump and flow control valves in series.
Cooling System	Beginner	Water (liquid)	20	2	Learn how to link GSC variables together to apply multiple variables and goals in a single simulation.
Model Calibration with Pipe Variables	Beginner	Water (liquid)	6	0	Learn how to use the GSC module to vary pipe friction to calibrate a model to match a measured volumetric flow rate.
Spray Discharge System	Intermediate	Water (liquid)	11	0	Learn how to create groups and use the Group Max/Min and Group Sum goals using the GSC module.
Controlled Heat Exchanger Temperature	Intermediate	Water (liquid)	10	1	Learn how to use the GSC module with heat transfer to vary the open percentage of a three-way valve to achieve a desired temperature.
NFPA Firewater System	Intermediate	Water (liquid)	71	0	Learn how to use the GSC module to determine the most hydraulically remote

Example	Complexity	Fluid	Pipes	Pumps	Description
					nozzle for NFPA Reports.

Beginner - Heat Transfer in a Pipe - GSC

Summary

The objective of this example is to demonstrate how goals and variables are used in the GSC module to achieve an objective.

For this example, we will use the GSC module to determine the flow rate required such that the fluid at the outlet of a 30-meter section of pipe, with heat transfer, reaches a temperature of 50 deg. C.

Note: This example can only be run if you have a license for the GSC module.

Topics Covered

- Defining the Goal Seek and Control group
- Understanding Goal Seek and Control Output

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *Metric - Beginner - Heat Transfer in a Pipe.fth*
- *Metric - Beginner - Heat Transfer in a Pipe - GSC.fth*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **Metric - Heat Transfer in a Pipe.fth** example file listed above, which is located in the Examples folder in the AFT Fathom application folder. Save the file to a different folder. The workspace should look like Figure 1 below:

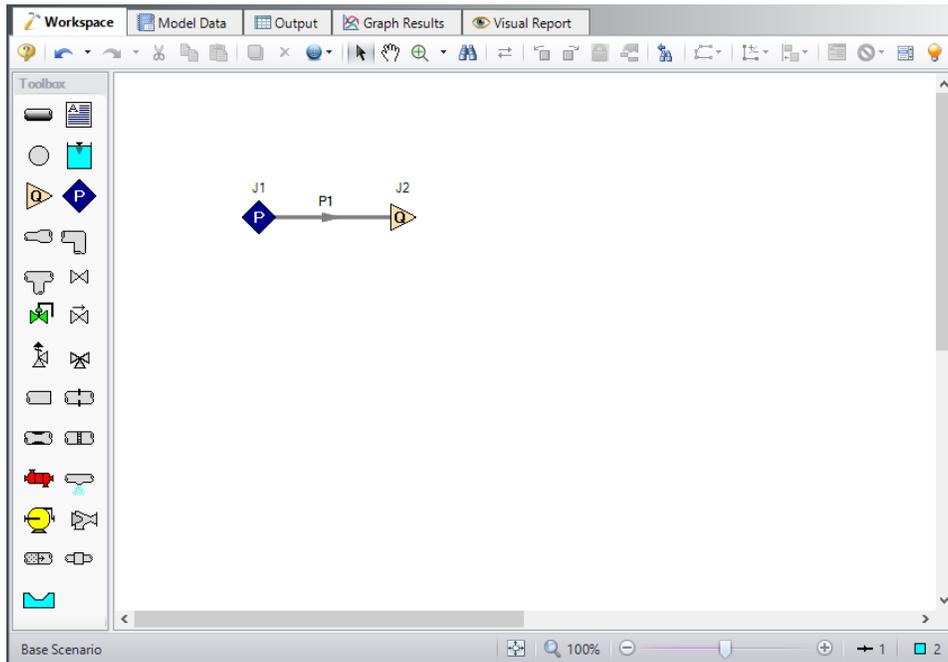


Figure 1: Workspace for GSC Heat Transfer example

Step 3. Define the Modules group

The GSC module is controlled through the Goal Seek and Control group in Analysis Setup. Open Analysis Setup and navigate to the Modules panel. Check the box next to Activate GSC. The Use option should automatically be selected, making GSC enabled for use. Alternatively, GSC can be enabled from the Analysis menu by going to the Goal Seek & Control option and selecting Use, as shown in Figure 2.

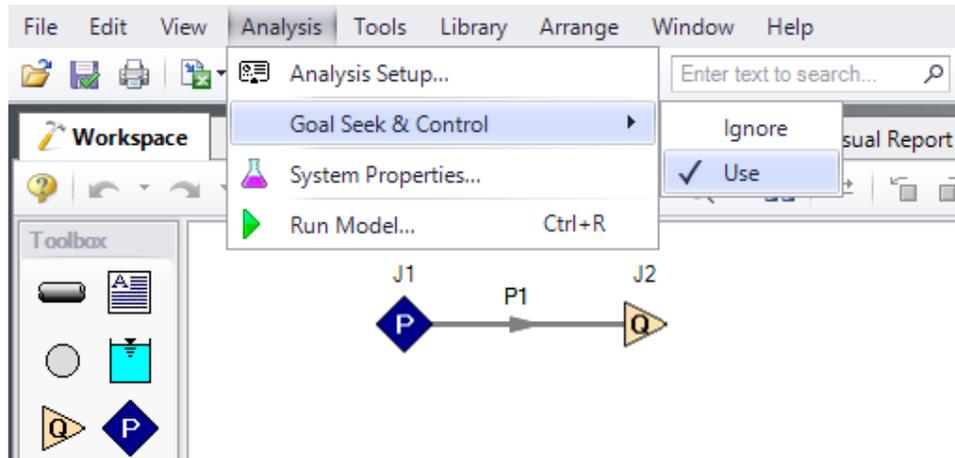


Figure 2: GSC can be enabled and disabled from the Analysis menu

Step 4. Define the Goal Seek and Control Group

The Goal Seek and Control group is made up of three items: Variables, Goals, and Numerical Controls. The default settings in Numerical Controls will suffice for most applications and no input is necessary in this panel for this example. The variables and goals are defined in their respective panels. The Variables panel is shown in Figure 3 below:

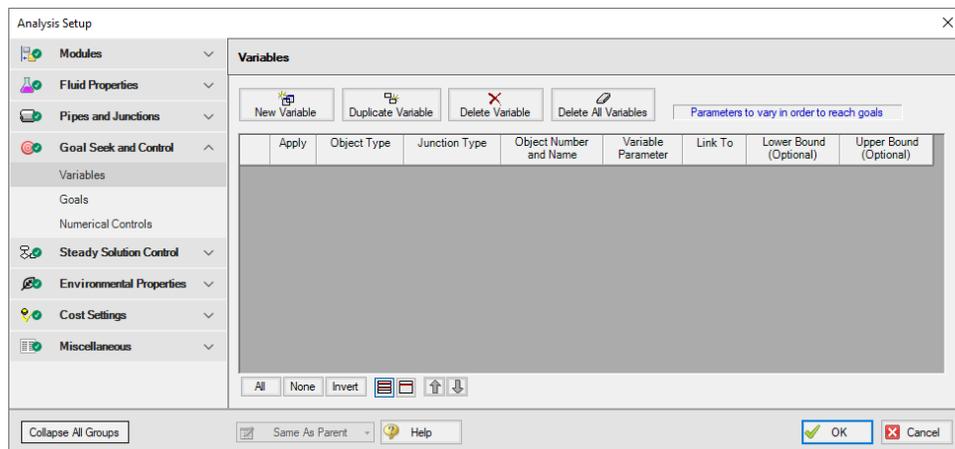


Figure 3: Variables panel

Variables Panel

In the GSC module, variables are the parameters that the solver will modify in order to achieve the specified goals. There should be one variable defined for each goal.

Goal Seek and Control (GSC) Examples

Open the Variables panel in the Goal Seek and Control group in Analysis Setup. The Variables panel allows you to create and modify the system variables. The object type and junction type are selected, then the name and number of the object to which the variable applies, and the object parameter that is to be varied are specified all on the Variables panel.

To create a new variable, follow these steps:

1. Click New Variable
2. Apply = Checked
3. Object Type = Junction
4. Junction Type = Assigned Flow
5. Junction Number and Name = J2 (Assigned Flow)
6. Variable Parameter = Flow
7. Link To = (None)
8. Lower Bound (Optional) = 0

The Apply column allows users to specify which of the variables that have been defined will be used. This allows the flexibility of creating multiple variable cases, while only applying selected variables for any given run.

The Link To column allows users to apply the same variable to multiple objects. This allows users to force parameters for several objects to be varied identically.

Upper and Lower Bounds provide logical extremes during the goal search. For this case, leave both the lower bound and the upper bound values blank.

After entering the data, the Variables panel should appear as shown in Figure 4:

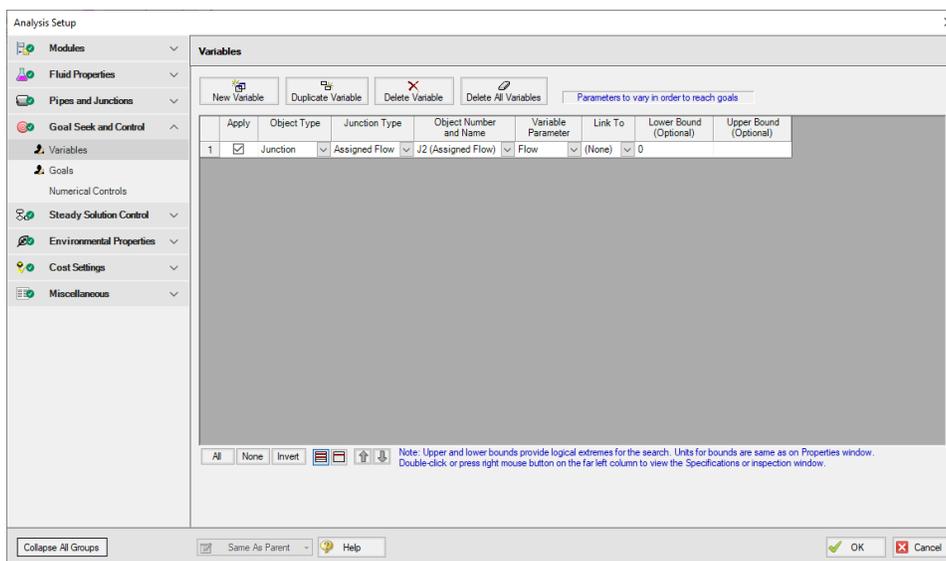


Figure 4: The Variables panel is used to specify the variables

Goals Panel

Goal Seek and Control (GSC) Examples

Goals are the parameter values set by the user that Fathom adjusts the variables to achieve. There should be one goal defined for each variable.

Open the Goals panel in the Goal Seek and Control group. The Goals panel allows you to create and modify the system goals. The goal type, object type, and the goal parameter are selected. A criterion for determining if the goal has been met is then specified, along with a value and units for the goal parameter. You can select the object to which the goal applies, and, if applicable, the location on the object at which the goal applies (i.e. the inlet or outlet of a pipe object).

Do the following:

1. Click New Goal
2. Apply = Checked
3. Goal Type = Point
4. Object Type = Pipe
5. Goal Parameter = Temperature
6. Criteria = =
7. Goal Value = 50
8. Goal Units = deg. C
9. Object ID = P1 (Pipe)
10. Object Location = Outlet

After entering the data, the Goals panel should appear as shown in Figure 5:

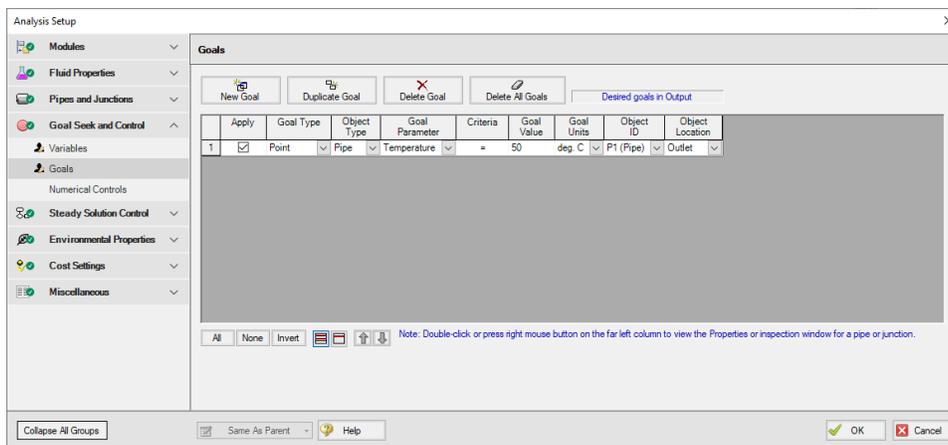


Figure 5: The Goals panel is used to specify the goals

As variables and goals are added to a model, Fathom will display symbols beside the pipes and junctions that have variables or goals applied to them. The default is a V for variables, and a G for goals. The goal symbol is not displayed next to objects that are part of a group goal. This is illustrated in Figure 6:

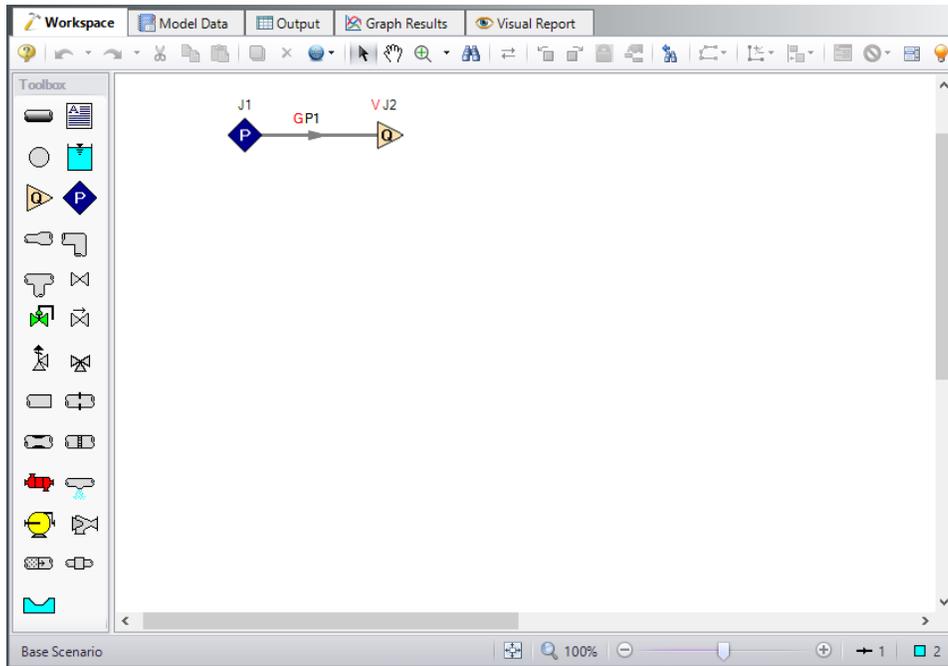


Figure 6: Workspace with Goal and Variable symbols displayed

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

Figure 7 shows the Output window with the General, Pipes, and Junctions sections displayed.

The screenshot shows the 'Output' window with three sections: General, Pipes, and Junctions. The 'General' section shows a table with one row of data. The 'Pipes' section shows a table with one row of data. The 'Junctions' section shows a table with two rows of data.

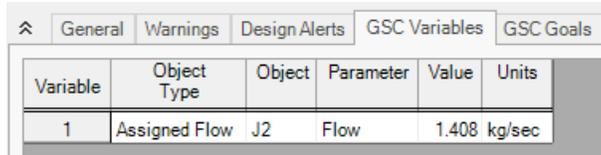
Variable	Object Type	Object	Parameter	Value	Units
1	Assigned Flow	J2	Flow	1.409	kg/sec

Pipe	Name	Vol. Flow Rate (m3/hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Heat Flux (kW/m2)	Heat Rate (kW)	T Inlet (deg. C)	T Outlet (deg. C)
1	Pipe	5.145	0.07668	3.450	3.449	0	0	4.793E-04	4.793E-04	0	4.960E-03	3.450	3.449	3.450	3.450	-1.823	-88.22	65.00	50.00

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Assigned Pressure	3.450	3.450	3.450	3.450	5.145	1.409	0
2	Assigned Flow	3.449	3.449	3.450	3.450	5.145	1.409	0

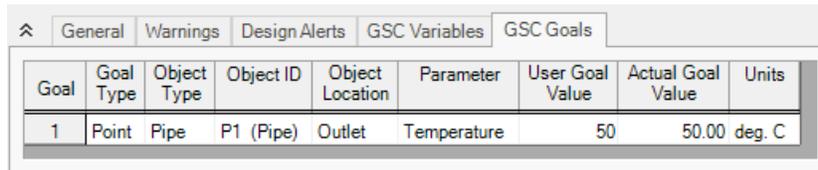
Figure 7: The Output window contains all the data that was specified in the output control window

The results of the GSC analysis are shown in the General Output section in the GSC Variables and GSC Goals tabs. The GSC Variables tab shows the final value for the variable parameter, as shown in Figure 8. The GSC Goals tab shows the final value achieved for the goal, as shown in Figure 9.



Variable	Object Type	Object	Parameter	Value	Units
1	Assigned Flow	J2	Flow	1.408	kg/sec

Figure 8: GSC Variable Results are displayed in the General section of the Output Window



Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units
1	Point	Pipe	P1 (Pipe)	Outlet	Temperature	50	50.00	deg. C

Figure 9: GSC Goal Results are displayed in the General section of the Output Window

The GSC module analysis determined that a mass flow rate of 1.408 kg/sec (5.143 m³/hr) results in a pipe outlet temperature of 50 deg. C. In order to achieve the desired temperature, the assigned flow rate was varied from the initial value of 4.5 kg/s, to the final value of 1.409 kg/sec. In addition to the GSC module tabs in the General Output section, the resulting pipe flow and temperature results can be seen in the Pipes section of the Output Window.

Pump Sizing and Selection with Flow Control Valves - GSC

Summary

The piping for a heat exchanger system is being designed. The system will pump supply water from a tank pressurized to 0.7 barG (70 kPa(g)) with a liquid surface elevation of 1.5 meters to a receiving tank with a pressure of 2 barG (200 kPa(g)) and a liquid surface elevation of 3 meters. The heat exchangers operate in parallel. The flow through the heat exchangers is controlled to 25 m³/hr by two flow control valves. There is a design requirement that the control valves have a minimum pressure drop of 0.35 bar (35 kPa).

Use the GSC module to size the pump.

Note: This example can only be run if you have a license for the GSC module.

Topics Covered

- Defining the Goal Seek and Control group
- Using Pump head rise as a GSC Variable
- Using Group Max/Min Goals

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through the [Beginner: Heat Transfer in a Pipe - GSC](#) example and the [Spray Discharge System - GSC](#) example, which demonstrates how to use Group Max/Min goals.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Pump Sizing and Selection with Flow Control Valves.fth***
- ***Metric - Pump Sizing and Selection with Flow Control Valves - GSC.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **Metric - Pump Sizing and Selection with Flow Control Valves.fth** example file and save it as a new file. Right click on the Base Scenario and select Create Child. Name the child scenario Goal Seek Control.

Set the J2 Pump Sizing Parameter to Head Rise, with a Fixed Head Rise of 6 meters. In this example, the head rise will become the GSC variable.

The workspace should look like Figure 1 below:

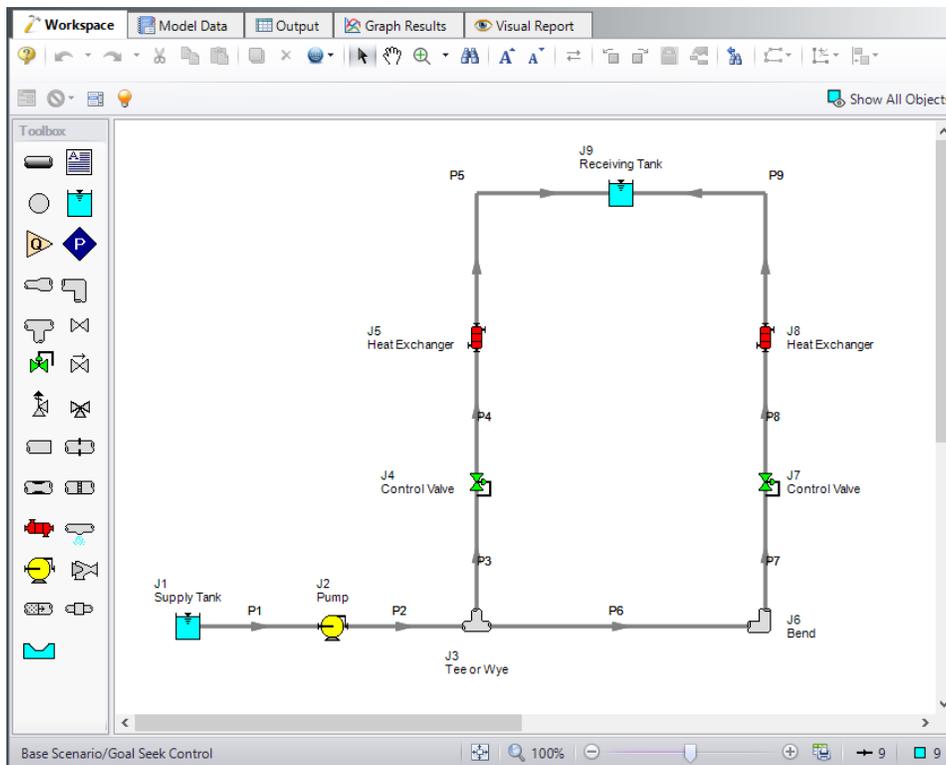


Figure 1: Workspace for GSC Pump Selection with Flow Control Valves example

Step 3. Create a Group

Create a group named *Flow Control Valves* by opening the Edit menu, hover over Groups, and select Create. Add both Control Valve junctions J4 and J7 to the group.

Step 4. Define the Modules Group

Open Analysis Setup, and navigate to the Modules panel. Check the box next to Activate GSC. The Use option should automatically be selected, making GSC enabled for use.

Step 5. Define the Goal Seek and Control Group

Specify the variables and goals for the model.

Variables Panel

Select the Variables panel in the Goal Seek and Control group. For this example, click New Variable and enter the following variable data:

1. Apply = Checked
2. Object Type = Junction
3. Junction Type = Pump
4. Junction Number and Name = J2 (Pump)
5. Variable Parameter = Head Rise
6. Link To = (None)

After entering the data, the Variables panel should appear as shown in Figure 2:

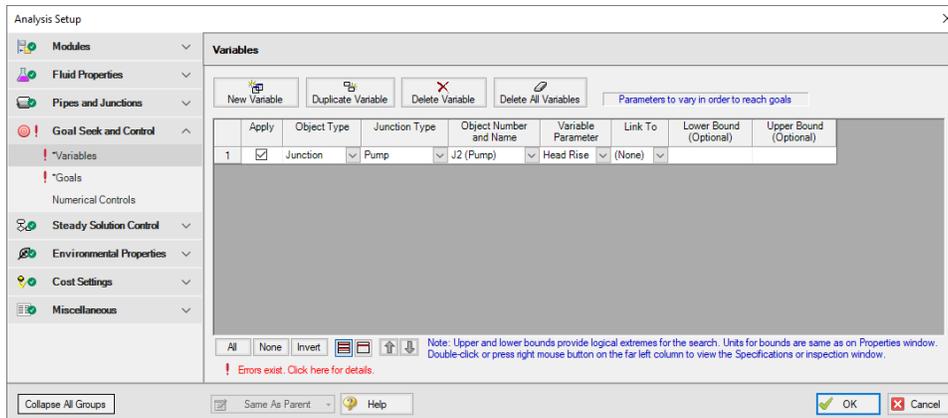


Figure 2: The Variables panel is used to specify the variables

Goals Panel

Select the Goals panel in the Goal Seek and Control group.

Create a new goal for the pressure drop across the flow control valves as defined below (if the option for a group goal is not visible, you may have forgotten to create a group as discussed above). Click New Goal and enter the following:

1. Apply = Selected
2. Goal Type = Group
3. Object Type = Group Max/Min
4. Goal Parameter = Pressure Loss
5. Criteria = >=
6. Goal Value = 0.35
7. Goal Units = bar
8. Object ID = Flow Control Valves
9. Object Location = NA

After entering the data, the Goals panel should appear as shown in Figure 3:

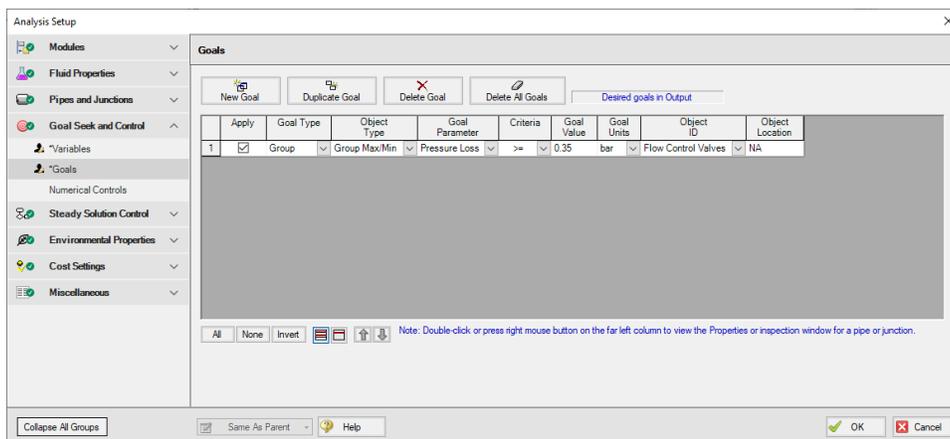


Figure 3: The Goals panel is used to specify the goals

Step 6. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 7. Examine the Output

The results of the GSC module analysis are shown in the General Output section. The GSC Variables tab shows the final value for the variable parameter, as shown in Figure 4. The pump head rise required for this system to supply a minimum of 0.35 bar (35 kPa) pressure drop at each of the Flow Control Valve junctions is 55.35 meters. The GSC Goals tab shows the final value achieved for the goal, as shown in Figure 5. This shows the goal of a minimum 0.35 bar (35 kPa) pressure drop was obtained for both valves.

Note that the results for this example are same as the results for the original Pump Selection with Flow Control Valves example.

Variable	Object Type	Object	Parameter	Value	Units
1	Pump	J2	Head Rise	55.35	meters

Figure 4: GSC Variable Results are displayed in the General section of the Output Window

Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units	Object Used for Group Goal
1	Group	Group Max/Min	Flow Control Valves	N/A	Pressure Loss	0.35	0.3500	bar	J7 (Control Valve)

Figure 5: GSC Goal Results are displayed in the General section of the Output Window

Cooling System - GSC

Summary

The objective of this example is to use the GSC module to vary the required pump speed and control pressure such that a minimum flow of 35 m³/hr is maintained through the Heat Exchanger, and the control valve open percentage remains above 30%.

Note: This example can only be run if you have a license for the GSC module.

Topics Covered

- Applying multiple variables and goals.
- Using linked variables

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through the [Beginner - Heat Transfer in a Pipe - GSC](#) example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Cooling System.fth***
- ***Metric - Cooling System - GSC.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the Model

Open the **Metric - Cooling System.fth** example file and save it as a new file.

Load the Pump A scenario and turn off Pump J5 by opening the properties window, navigating to the Optional tab, and selecting the Pump Off No Flow special condition.

Open the properties window for Control Valves J11. On the Control Valve Model tab, under Loss When Fully Open, select Cv and then select From Open % Table (on Optional Tab) for the Loss Source.

Go to the Optional tab and add the following data to Open Percentage Data table:

Open Pct.	Cv
0	0
100	150

Click OK. Make these same changes to Control Valves J12 and J13. The workspace should look like Figure 1 below:

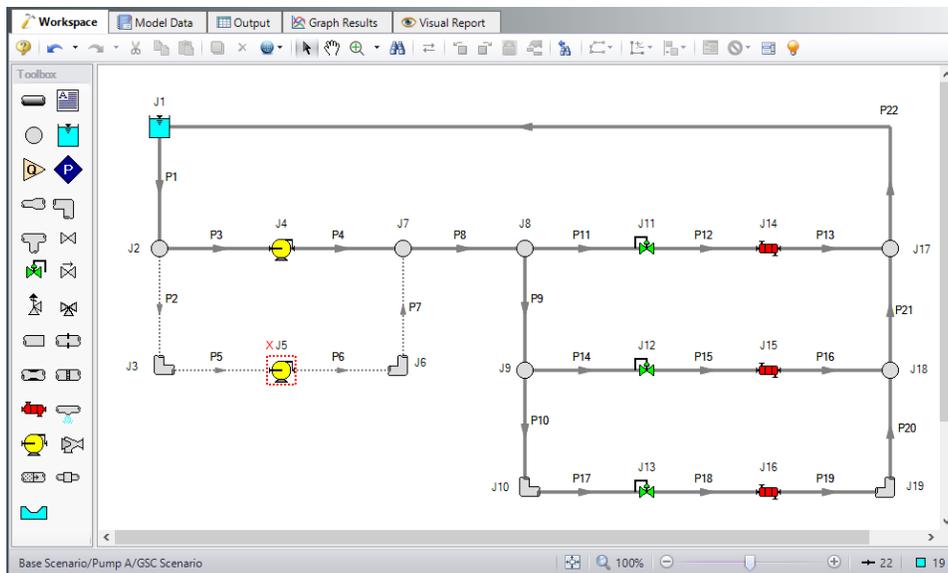


Figure 1: Workspace for GSC Cooling System example

Step 3. Define the Modules Group

Open Analysis Setup, and navigate to the Modules panel. Check the box next to Activate GSC. The Use option should automatically be selected, making GSC enabled for use.

Step 4. Define the Goal Seek and Control Group

Specify the variables and goals for the model.

Variables Panel

Open the Variables panel in the Goal Seek and Control group in Analysis Setup. For this example, we will be adding four variables: one pump variable and three control valve variables which will be linked. Enter the variable inputs shown in Figure 2:

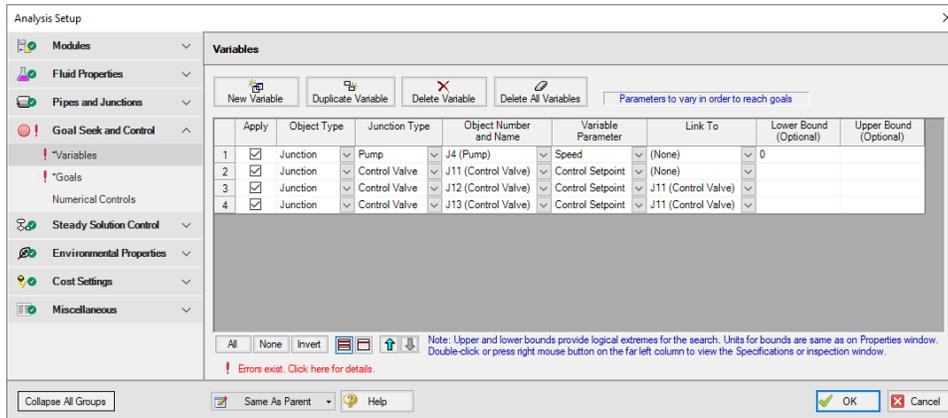


Figure 2: The Variables panel is used to specify the variables

Goals Panel

There should be one goal defined for each variable. In this case we essentially have two variables, since all of the control valve variables are linked, causing them to act as a single variable.

Open the Goals panel in the Goal Seek and Control group in Analysis Setup. Create two new goals as defined below in Figure 3:

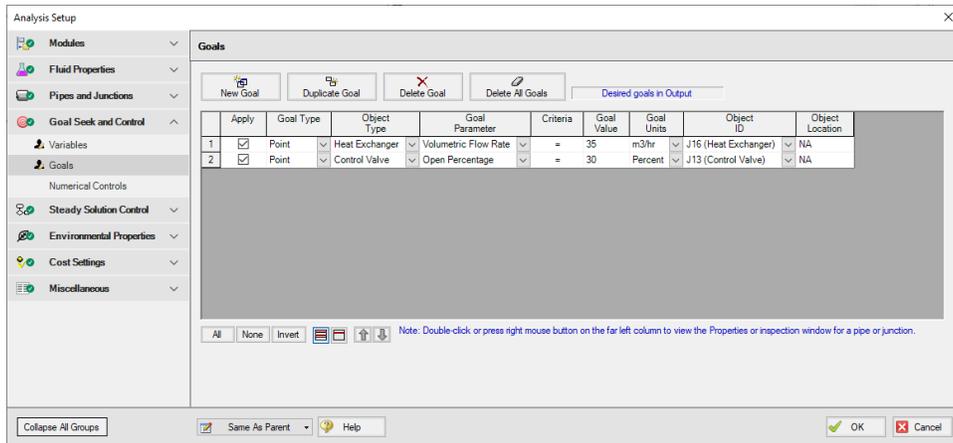


Figure 3: GSC Manager Goals tab is used to specify the goals

After entering the data, the Goals panel should appear as shown in Figure 3. Note that by setting the most hydraulically remote heat exchanger and control valve equal to the minimum requirements, we are ensuring that the other heat exchangers and control valves will exceed the minimum values.

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

The results of the GSC module analysis are shown in the General Output section. The GSC Variables tab shows the final values for the variable parameters, as shown in Figure 4. The pump speed was determined to be 84.26%, and the control setpoint for the valves was determined to be 3.094 barG (309.4 kPa(g)). The GSC Goals tab shows the final values achieved for the goals, as shown in Figure 5. Viewing the Valve Summary tab will show that control valves J11 and J12 have open percents above the minimum of 30%. The Heat Exchanger Summary tab will similarly show that heat exchangers J14 and J15 have volumetric flow rates above the required minimum of 35 m3/hr.

Reservoir Summary					
GSC Variables					
Variable	Object Type	Object	Parameter	Value	Units
1	Pump	J4	Speed	84.26	Percent
2	Control Valve	J11	Control Setpoint	3.094	barG
3	Control Valve	J12	Control Setpoint	3.094	barG
4	Control Valve	J13	Control Setpoint	3.094	barG

Figure 4: GSC Variable Results are displayed in the General section of the Output Window

GSC Goals								
Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units
1	Point	Heat Exchanger	J16 (Heat Exchanger)	N/A	Volumetric Flow Rate	35	35.05	m3/hr
2	Point	Control Valve	J13 (Control Valve)	N/A	Open Percentage	30	29.97	Percent

Figure 5: GSC Goal Results are displayed in the General section of the Output Window

Model Calibration with Pipe Variables - GSC

Summary

A supply reservoir feeds a water treatment pond through gravity flow and the pipeline has been in existence for about 15 years. The original pipeline delivered about 457 m³/hr and recently, the operators working at the treatment pond have reported that the flow has dropped by about 20%. Their lead engineer has determined that the cause of the flow reduction is most likely due to pipe corrosion.

The objective of this example is to demonstrate how to use the GSC module to determine the amount of additional frictional resistance in the pipeline due to years of corrosion. This could factor into the decision on whether or not to replace the pipeline and when.

For this example, the GSC module will be used to approximate the absolute roughness due to corrosion in the pipeline which has reduced the flow rate from 457 m³/hr to 366 m³/hr.

Note: This example can only be run if you have a license for the GSC module.

Topics Covered

- Defining the Goal Seek and Control group
- Setting Variables and Goals
- Understanding Goal Seek and Control Output

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through the [Beginner - Heat Transfer in a Pipe - GSC](#) example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Model Calibration with Pipe Variables - GSC.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = AFT Standard
 - b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
 - c. Temperature = 21 deg. C

Step 3. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

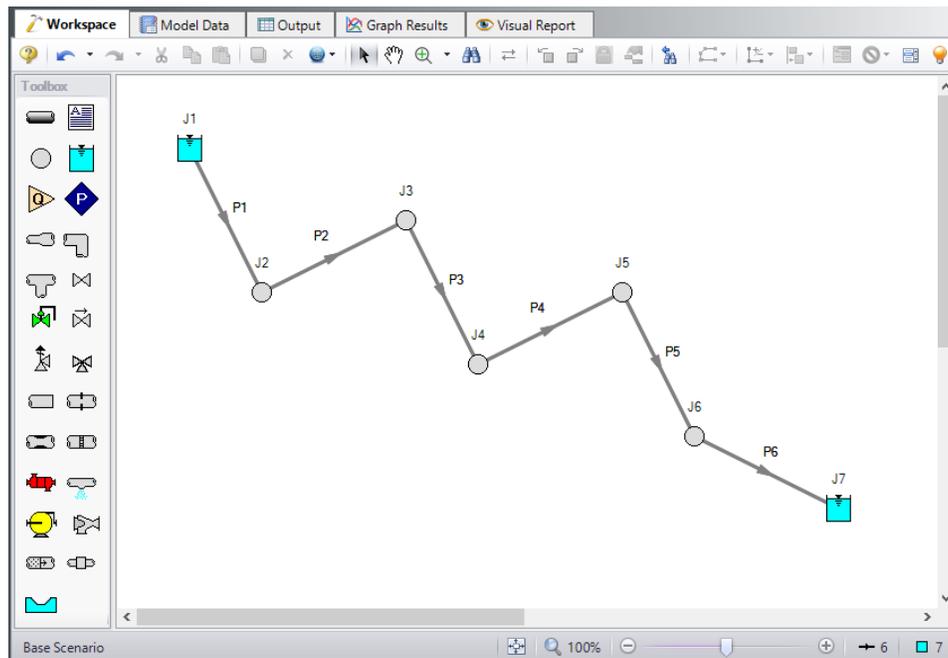


Figure 1: Workspace for GSC Model Calibration Using Pipe Variables Example

Enter the following pipe and junction properties.

Pipe Properties

1. All Pipes
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 8 inch
 - d. Type = STD (schedule 40)
 - e. Friction Model Data Set = Standard
 - f. Length =

Pipe	Length (meters)
1 & 5	1610
2, 3, & 4	805
6	610

Junction Properties

1. J1 Reservoir
 - a. Name = Supply Reservoir
 - b. Liquid Surface Elevation = 370 meters
 - c. Liquid Surface pressure = 0 barG (0 kPa(g))
 - d. Pipe Elevation = 305 meters
2. J2 Branch
 - a. Elevation = 150 meters
3. J3 Branch
 - a. Elevation = 180 meters
4. J4 Branch
 - a. Elevation = 30 meters
5. J5 Branch
 - a. Elevation = 90 meters
6. J6 Branch
 - a. Elevation = 15 meters
7. J7 Reservoir
 - a. Name = Treatment Pond
 - b. Liquid Surface Elevation = 3 meters
 - c. Liquid Surface Pressure = 0 barG (0 kPa(g))
 - d. Pipe Depth = 0 meters

To show the names of the reservoirs a new Workspace Layer will be created.

1. Create a new layer by selecting New Layer and then Blank Layer from the Workspace Layers tab of the Quick Access Panel.
2. Name the layer Reservoir Labels.
3. Edit the layer by selecting it and clicking the gear icon to bring up the Layer Settings window.
4. Open the Show/Hide Labels panel and uncheck Force Shown Labels to Match Shown Objects.
5. Toggle the visibility icon next to the reservoirs to on. This will cause their labels to be shown.
6. Navigate to the Junction Parameters panel and expand the Commonly Used Junction Parameters list.
7. Double-click Junction Name to add it to the list on the right-hand side. This will add the Junction Name to the label of the junctions that were defined in the Show/Hide Labels panel.
8. Close the Layer Settings window.

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 4. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 5. Examine the Output

After running the model based upon the original pipeline operation with brand new clean steel pipe, we find that the flow rate is 456.7 m³/hr, as shown in Figure 2.

Pipe	Name	Vol. Flow Rate (m ³ /hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)
1	Pipe	456.7	3.931	13.216	7.3023	305.00	150.000	-5.914	-5.914	-15.181	94.61	7.302	13.2165	7.379	13.294
2	Pipe	456.7	3.931	13.216	5.6449	150.00	180.000	7.572	7.572	2.938	47.31	13.216	5.6449	13.294	5.722
3	Pipe	456.7	3.931	15.703	5.6449	180.00	30.000	-10.058	-10.058	-14.691	47.31	5.645	15.7027	5.722	15.780
4	Pipe	456.7	3.931	15.703	5.1929	30.00	90.000	10.510	10.510	5.876	47.31	15.703	5.1929	15.780	5.270
5	Pipe	456.7	3.931	5.193	3.2718	90.00	15.000	1.921	1.921	-7.346	94.61	5.193	3.2718	5.270	3.349
6	Pipe	456.7	3.931	3.272	0.9361	15.00	3.000	2.336	2.336	-1.175	35.85	3.272	0.9361	3.349	1.013

Figure 2: Original pipeline results based upon clean pipe

Step 6. Create a child scenario named Flow Reduction from Corrosion

In the child scenario, we are going to use the GSC module to determine how much additional frictional resistance due to corrosion is in the pipeline. In this new scenario, we know that the flow rate has been reduced to 366 m³/hr in the physical system by corrosion and therefore the GSC goal in this case is going to be the volumetric flow rate.

The GSC module is a very useful tool that allows one to calibrate their models so that the predicted results can match measured data. There are four different GSC pipe variables that can be used to assist with this calibration including the specified pipe Friction (e.g. absolute roughness), pipe Design Factor, pipe ID (scaling), and Insulation Thickness.

In order to use GSC to determine the amount of additional frictional resistance due to corrosion, an input parameter must be varied. In this case, we are going to vary each pipe's absolute roughness in order to achieve the desired goal flow rate of 366 m³/hr.

Step 7. Define the Modules Group

Open Analysis Setup, and navigate to the Modules panel. Check the box next to Activate GSC. The Use option should automatically be selected, making GSC enabled for use.

Step 8. Define the Goal Seek and Control Group

Specify the variables and goals for the model.

Variables Panel

Select the Variables panel within the Goal Seek and Control group. The Variables panel allows users to create and modify the system variables.

In this scenario, we are going to vary the current pipe friction model (i.e. absolute roughness) for each pipe in the system. Since there are six individual pipes, there will be six individual pipe variables. If all six variables were to be able to vary independently of each other and there is only one goal associated, then this could lead to a less unique solution (as there could be several different combinations of pipe roughness values that would lead to the desired result). Therefore, we are going to assume that the entire pipeline has the same level of additional frictional resistance due to corrosion. In order to accomplish this, all pipe variables can be linked together. Since only one goal is present, we will only need one effective variable and so all of the pipe variables will be linked to the first variable.

Since there are six pipes, we will set pipes P2 - P6 to be linked to pipe P1.

For pipe P1, set the following variable data:

1. Apply = Selected
2. Object Type = Pipe
3. Junction Type = NA
4. Object Number and Name = P1 (Pipe)
5. Variable Parameter = Friction
6. Link To = (None)

Now, add new variables for pipes P2 - P6. The only differences between the variables for these pipes and the variable for pipe P1 will be the Object Number and Name and instead of selecting (None) under the Link To column, select P1 (Pipe). This will link pipes P2 - P6 to pipe P1 which will force all six pipe

variables to be varied identically, requiring only one goal to be specified. Figure 3 below shows all the linked pipe variables.

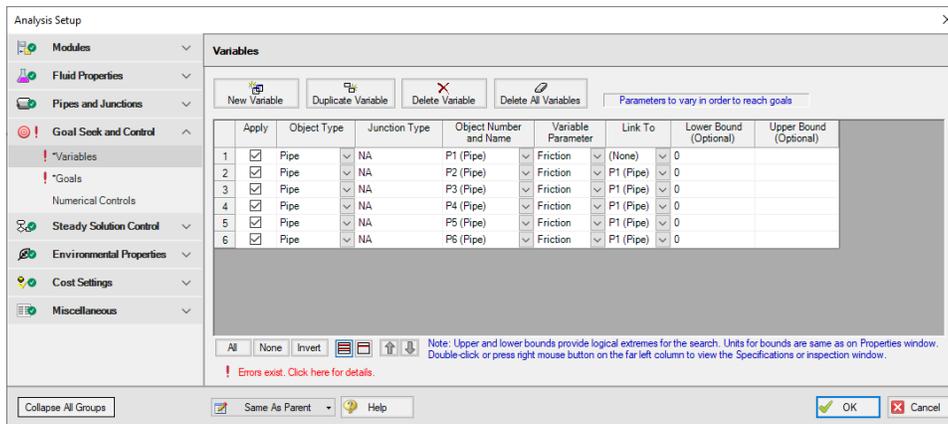


Figure 3: Six linked pipe variables (working effectively as a single variable) to vary the pipe friction model (i.e. absolute roughness)

Goals Panel

Goals are the parameter values set by the user that the solver tries to achieve by adjusting the variables. There should be one goal defined for each unlinked (and thus independent) variable. Since variables 2 - 6 are all linked to variable 1, only one goal will be required.

Select the Goals panel in the Goal Seek and Control group.

Create a new goal and set up the following goal information:

1. Apply = Selected
2. Goal Type = Point
3. Object Type = Pipe
4. Goal Parameter = Volumetric Flow Rate
5. Criteria = =
6. Goal Value = 366
7. Goal Units = m3/hr
8. Object ID = P6 (Pipe)
9. Object Location = NA

After entering the data, the Goals panel should appear as shown in Figure 4.

On the Numerical Control panel, the Search Method can be modified for Goals between two options, Beta Method and Least Squares. The Beta Method is useful in situations where there are multiple variables and multiple goals. For this model, make sure that the Least Squares method is selected.

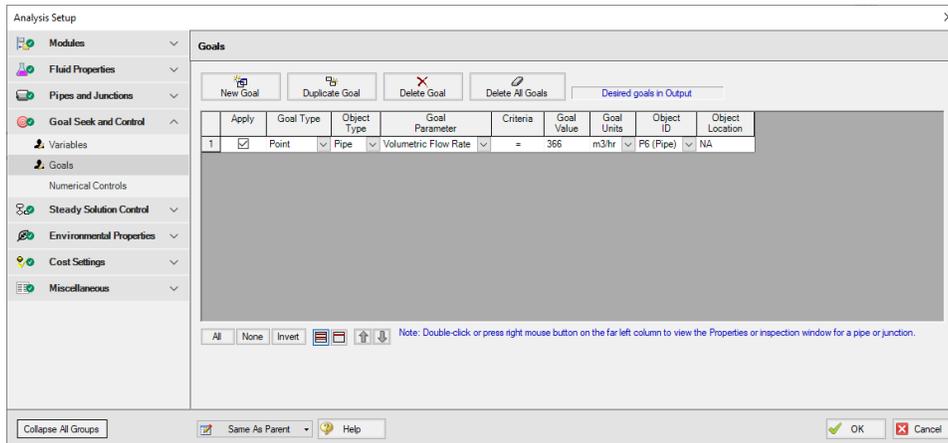


Figure 4: GSC Manager Goals tab is used to specify the goals

Step 9. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 10. Examine the Output

The results of the GSC module analysis are shown in the General section of the Output window. The GSC Variables tab shows the final value for the variable parameter, as shown in Figure 5. The resulting absolute roughness value that would decrease the flow rate to 366 m3/hr would be about 0.03966 cm. The additional frictional resistance in the pipeline due to corrosion is about nine times larger than the absolute roughness for clean steel pipe. The GSC Goals tab shows the final value achieved for the goal, as shown in Figure 6.

Variable	Object Type	Object	Parameter	Value	Units
1	Pipe	P1	Friction	0.03966	cm
2	Pipe	P2	Friction	0.03966	cm
3	Pipe	P3	Friction	0.03966	cm
4	Pipe	P4	Friction	0.03966	cm
5	Pipe	P5	Friction	0.03966	cm
6	Pipe	P6	Friction	0.03966	cm

Figure 5: GSC Variable Results are displayed in the General section of the Output Window

Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units
1	Point	Pipe	P6 (Pipe)	N/A	Volumetric Flow Rate	366	366.0	m3/hr

Figure 6: GSC Goal Results are displayed in the General section of the Output Window

Spray Discharge System - GSC

Summary

The objective of this example is to demonstrate how the Group Max/Min and Group Sum goal types can be used in the GSC module to achieve a goal for a group of objects.

For this example, we will use the GSC module to solve for the supply pressure required to:

Supply a minimum of 30 m³/hr to each spray discharge junction using a Group Max/Min goal.

Supply a total flow rate of 230 m³/hr through the system using a Group Sum goal.

Note: This example can only be run if you have a license for the GSC module.

Topics Covered

- Creating Groups
- Using Group Max/Min and Group Sum Goals

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through the [Beginner - Heat Transfer in a Pipe - GSC](#) example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Spray Discharge System.fth***
- ***Metric - Spray Discharge System - GSC.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the Model

Open the **Metric - Spray Discharge System.fth** example file and save it as a new file. We will not need the existing child scenarios for this example: right click on the Base Scenario in the Scenario Manager and select Delete All Children. The workspace should look like Figure 1 below:

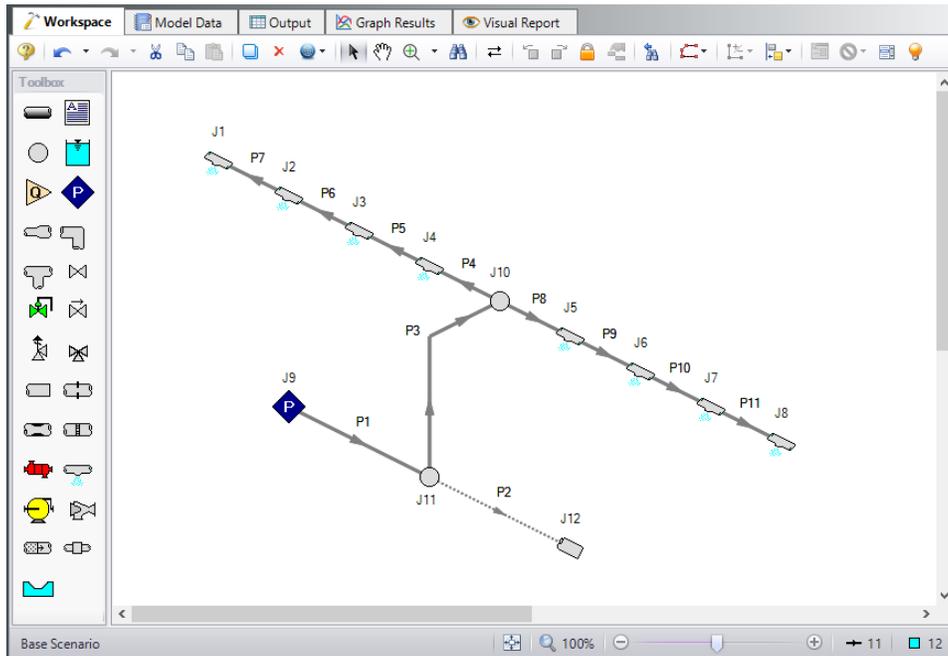


Figure 1: Workspace for GSC Spray Discharge system example

Step 3. Create a Group

In order to use the Group Max/Min and Group Sum goal types, the junctions included in the goal must be added to a group. To create a group, go to the Edit menu and select the Groups -> Create option. Give the new group the name Spray Discharges, and click OK. The Group Manager will be displayed. Select all of the Spray Discharge junctions from the junction list, as shown in Figure 2. After choosing the Spray Discharge junctions, click OK on the Group Manager form.

Note: If the Spray Discharge junctions were selected in the Workspace before Create Group was selected from the Edit menu, then they would be automatically selected for us in the Group Manager window.

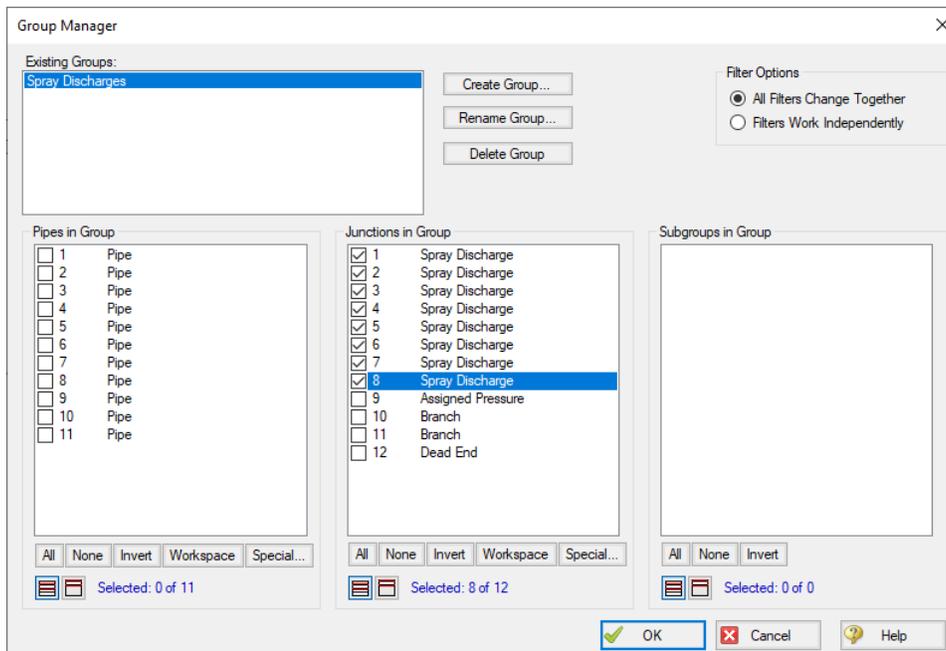


Figure 2: Groups are Created Using the Group Manager

Step 4. Define the Modules Group

Open Analysis Setup, and navigate to the Modules panel. Check the box next to Activate GSC. The Use option should automatically be selected, making GSC enabled for use.

Step 5. Define the Goal Seek and Control Group

Specify the variables and goals for the model.

Variables Panel

Open the Variables panel in the Goal Seek and Control group. For this example select New Variable, then choose following variable data:

1. Apply = Selected
2. Object Type = Junction
3. Junction Type = Assigned Pressure
4. Junction Number and Name = J9 (Assigned Pressure)
5. Variable Parameter = Pressure
6. Link To = (None)

After entering the data, the Variable panel should appear as shown in Figure 3:

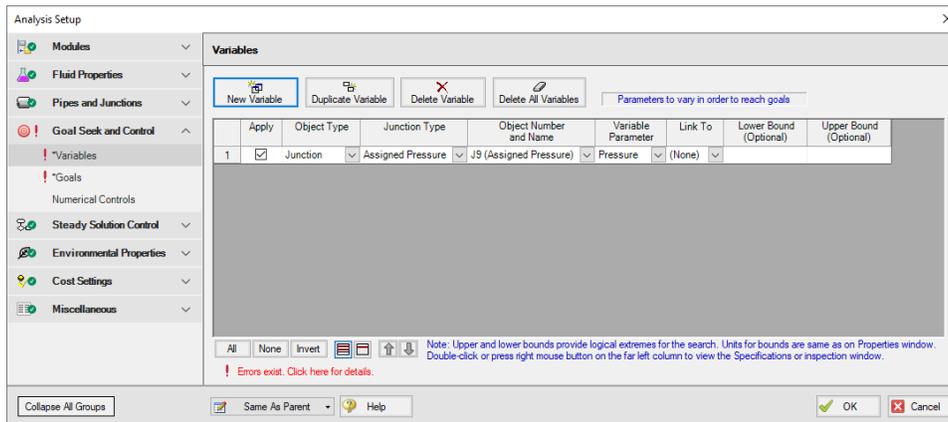


Figure 3: The Variables panel is used to specify the variables

Goals Panel

Open the Goals panel next. For this example, we are going to define two goals. We will apply one goal at a time for the sake of comparing the impact that each goal has on the results.

The first goal type is a Group Max/Min goal. This type of goal allows a single goal to be applied to a group of objects. Fathom applies a Group Max/Min goal by ensuring the final goal value is either greater than or equal to (Min) or less than or equal to (Max) the specified value. For this example, a Group Max/Min goal will be applied to ensure the minimum flow rate out of all of the spray discharge junctions is at least 30 m³/hr.

The second goal type is a Group Sum goal. This type of goal is used to set the sum of the goal parameter for all the members in the group to a specified value. For this example, the Group Sum goal will be applied to ensure the sum of the flow rates through the spray discharges equals 230 m³/hr.

Create two new goals defined as follows:

Group Max/Min:

1. Apply = Checked
2. Goal Type = Group
3. Object Type = Group Max/Min
4. Goal Parameter = Volumetric Flow Rate Out
5. Criteria = >=
6. Goal Value = 30
7. Goal Units = m³/hr
8. Object ID = Spray Discharges
9. Object Location = NA

Group Sum:

1. Apply = Not Checked
2. Goal Type = Group
3. Object Type = Group Sum
4. Goal Parameter = Volumetric Flow Rate
5. Criteria = =
6. Goal Value = 230
7. Goal Units = m3/hr
8. Object ID = Spray Discharges
9. Object Location = NA

After entering the data, the Goals panel should appear as shown in Figure 4 in Step 7.

Step 6. Create Scenarios

Using the Scenario Manager on the Quick Access Panel, create two child scenarios, one named *Group Max Min* and another named *Group Sum*.

Step 7. Select Group Max/Min Goal

Load the Group Max Min scenario.

Open the Goals panel in the Goal Seek and Control group in Analysis Setup. Make sure the Group Max/Min goal is applied, indicated by the check mark in the Apply column for that goal. Make certain Group Sum goal is NOT applied by unchecking the check box in the Apply column. The GSC goals should appear as shown in Figure 4:

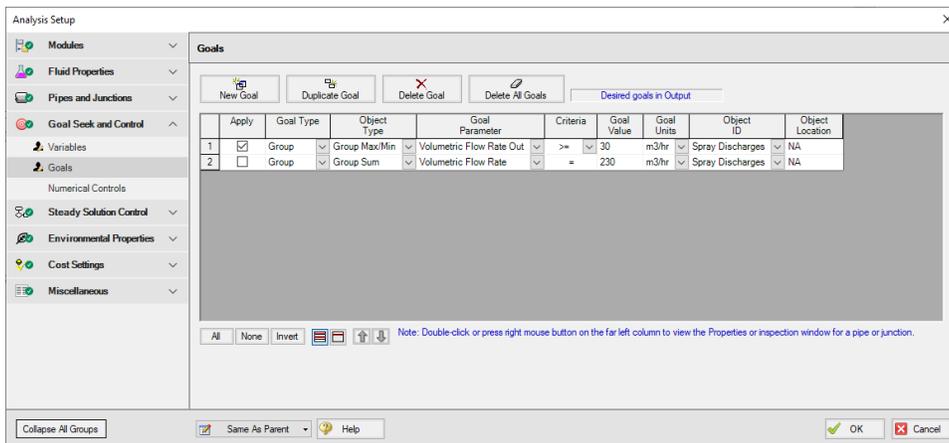


Figure 4: Group Max Min Goal Applied in the Goals panel

Step 8. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 9. Examine the Output

The GSC Variables tab shows the final value for the variable parameter, as shown in Figure 5. The supply pressure required to supply a minimum of 30 m³/hr at each of the Spray Discharge junctions is 18.85 barG (1,885 kPa(g)). The GSC Goals tab shows the value achieved for the goal, as shown in Figure 6. The flow rates determined for the Spray discharge junctions are shown in Figure 7. The spray discharge flow rates shown in Figure 7 illustrate how the Group Max/Min resulted in a the flow rate through each of the spray discharges meeting or exceeding the specified minimum flow rate of 30 m³/hr. The total system flow is 262.05 m³/hr.

Variable	Object Type	Object	Parameter	Value	Units
1	Assigned Pressure	J9	Pressure	18.85	barG

Figure 5: GSC Variable Results are displayed in the General section of the Output Window

Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units	Object Used for Group Goal
1	Group	Group Max/Min	Spray Discharges	N/A	Volumetric Flow Rate Out	30	30.00	m ³ /hr	J1 (Spray Discharge)

Figure 6: GSC Goal Results are displayed in the General section of the Output Window

Spray Discharge System - GSC

Spray Discharge	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Spray Discharge	8.676	1.013	8.877	1.013	30.00	8.323	39.12
2	Spray Discharge	N/A	1.013	9.189	1.013	N/A	N/A	39.12
3	Spray Discharge	N/A	1.013	10.433	1.013	N/A	N/A	39.12
4	Spray Discharge	N/A	1.013	13.366	1.013	N/A	N/A	39.12
5	Spray Discharge	N/A	1.013	13.366	1.013	N/A	N/A	39.12
6	Spray Discharge	N/A	1.013	10.433	1.013	N/A	N/A	39.12
7	Spray Discharge	N/A	1.013	9.189	1.013	N/A	N/A	39.12
8	Spray Discharge	8.676	1.013	8.877	1.013	30.00	8.323	39.12

Figure 7: Flow Rates Calculated for the Spray Discharge junctions for Group Max/Min goal

Step 10. Select Group Sum Goal

Load the Group Sum scenario.

Open the Goals panel in the Goal Seek and Control group from Analysis Setup. Apply the Group Sum goal by clicking the check box next to the Group Sum goal in the Apply column. Make certain Group Max/Min goal is NOT applied by unchecking the check box in the Apply column. The GSC goals should appear as shown in Figure 8:

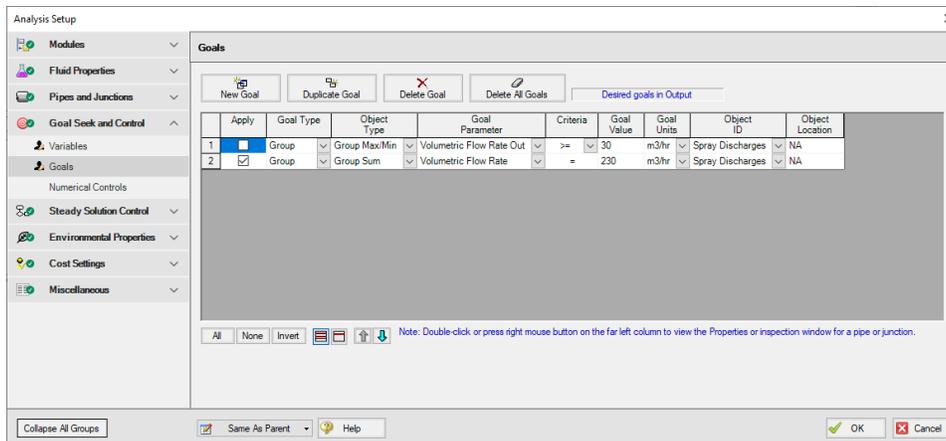


Figure 8: Group Sum Goals are applied in the GSC Manager

Step 11. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 12. Examine Output

The GSC Variables tab shows the final value for the variable parameter, as shown in Figure 9. The supply pressure required to supply a total system flow of 230 m³/hr is 14.60 barG (1,460 kPa(g)). The GSC Goals tab shows the final value achieved for the goal, as shown in Figure 10. The goal of 230 m³/hr for the system flow in this example was achieved. The flow rates determined for the Spray Discharge junctions are shown in Figure 11. Note that overall, the flows through the spray discharges in the Group Sum scenario are less than the flows determined for the Group Max/Min scenario. This is because in the Group Max/Min scenario, the restriction was placed on the flow rate at each of the spray discharges, resulting in a system flow of 262.05 m³/hr, while the restriction in the Group Sum scenario was on the total system flow equaling 230 m³/hr.

Variable	Object Type	Object	Parameter	Value	Units
1	Assigned Pressure	J9	Pressure	14.60	barG

Figure 9: GSC Variable Results are displayed in the General section of the Output Window

Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units
2	Group	Group Sum	Spray Discharges	N/A	Volumetric Flow Rate	230	230.0	m ³ /hr

Figure 10: GSC Goal Results are displayed in the General section of the Output Window

Spray Discharge	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m ³ /hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Spray Discharge	6.913	1.013	7.068	1.013	26.32	7.303	39.12
2	Spray Discharge	N/A	1.013	7.309	1.013	N/A	N/A	39.12
3	Spray Discharge	N/A	1.013	8.271	1.013	N/A	N/A	39.12
4	Spray Discharge	N/A	1.013	10.534	1.013	N/A	N/A	39.12
5	Spray Discharge	N/A	1.013	10.534	1.013	N/A	N/A	39.12
6	Spray Discharge	N/A	1.013	8.271	1.013	N/A	N/A	39.12
7	Spray Discharge	N/A	1.013	7.309	1.013	N/A	N/A	39.12
8	Spray Discharge	6.913	1.013	7.068	1.013	26.32	7.303	39.12

Figure 11: Flow Rates Calculated for the Spray Discharge junctions for Group Sum Goal

Controlled Heat Exchanger Temperature - GSC

Summary

The objective of this example is to use the GSC module to vary a Three Way Valve percent open to divert heat exchanger flow to maintain a downstream temperature of 50 deg. C for a specified upstream supply temperature.

Note: This example can only be run if you have a license for the GSC module.

Topics Covered

- Using GSC to control flow splits

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through the [Beginner - Heat Transfer in a Pipe - GSC](#) example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Controlled Heat Exchanger Temperature.fth***
- ***Metric - Controlled Heat Exchanger Temperature - GSC.fth***
- ***Controlled Heat Exchanger Temperature.dat***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the Model

Open the **Metric - Controlled Heat Exchanger Temperature.fth** example file and save it as a new file. The workspace should look like Figure 1 below:

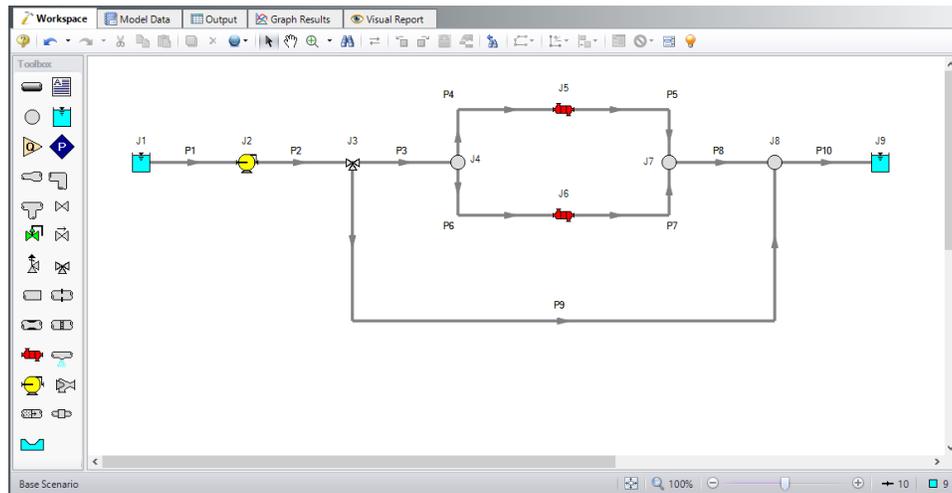


Figure 1: Workspace for GSC Controlled Heat Exchanger Temperature example

Step 3. Define the Modules Group

Open Analysis Setup, and navigate to the Modules panel. Check the box next to Activate GSC. The Use option should automatically be selected, making GSC enabled for use.

Step 4. Define the Goal Seek and Control Group

Specify the variables and goals for the model.

Variables Panel

Open the Variables panel in the Goal Seek and Control group. For this example, we will be adding a variable for the Three Way Valve % Open. Click New Variable and enter the following data:

1. Apply = Selected
2. Object Type = Junction
3. Junction Type = Three Way Valve
4. Junction Number and Name = J3 (Three Way Valve)
5. Variable Parameter = Open Percentage
6. Link To = (None)

7. Lower Bound = 0
8. Upper Bound = 100

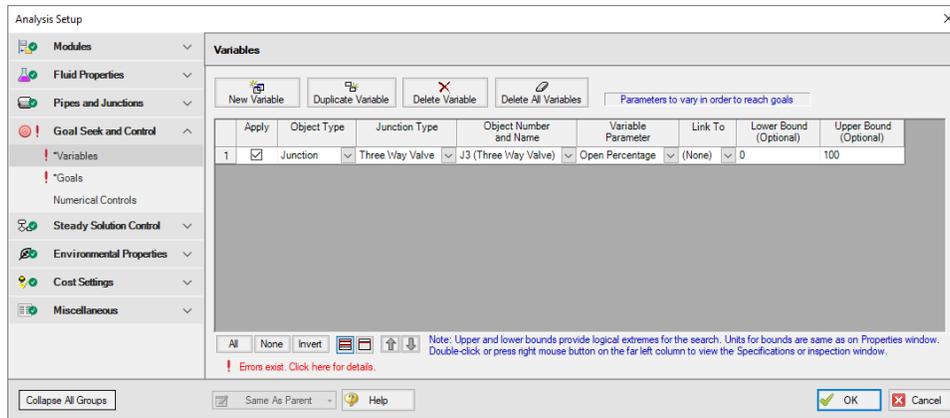


Figure 2: GSC Manager Variables tab is used to specify the variables

Goals Panel

Open the Goals panel in the Goal Seek and Control group in Analysis Setup. Click New Goal for the downstream temperature and define it with the data below:

1. Apply = Selected
2. Goal Type = Point
3. Object Type = Branch
4. Goal Parameter = Temperature
5. Criteria = =
6. Goal Value = 50
7. Goal Units = deg. C
8. Object ID = J8 (Branch)
9. Object Location = Outlet

After entering the data, the Goals panel should appear as shown in Figure 3:

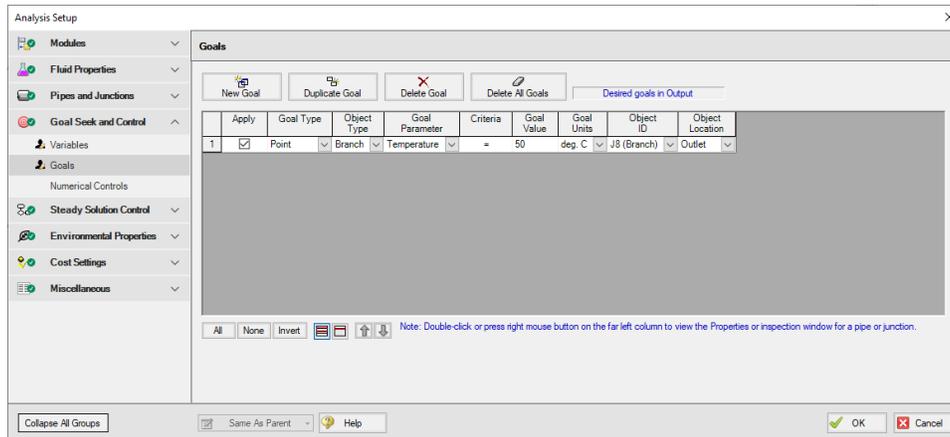


Figure 3: The Goals panel is used to specify the goals

Step 5. Run the Model

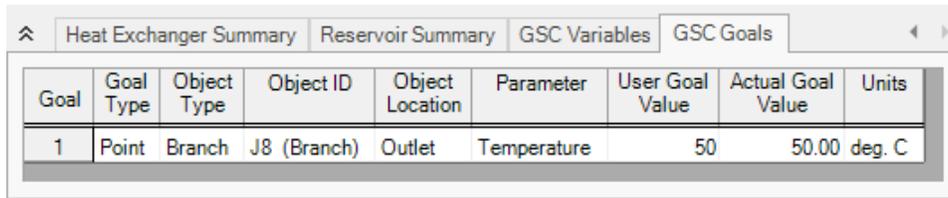
Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

The results of the GSC module analysis are shown in the General Output section. The GSC Variables tab shows the final value for the variable parameter, as shown in Figure 4. The open percentage required for the Three Way Valve to achieve a temperature of 50 deg. C at Branch J8 is 79.24%. The GSC Goals tab shows the final value achieved for the goal, as shown in Figure 5. This shows the goal of 50 deg. C was achieved for the downstream temperature.

Variable	Object Type	Object	Parameter	Value	Units
1	Three Way Valve	J3	Open Percentage	79.24	Percent

Figure 4: GSC Variable Results are displayed in the General section of the Output Window



The screenshot shows a software window titled "GSC Goals" with a tabbed interface. The active tab is "GSC Goals". Below the tabs is a table with the following data:

Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units
1	Point	Branch	J8 (Branch)	Outlet	Temperature	50	50.00	deg. C

Figure 5: GSC Goal Results are displayed in the General section of the Output Window

NFPA Firewater System - GSC

Summary

The goal of this example is to determine how to use the GSC module to determine the hydraulically most remote path to generate an NFPA report.

Note: This example can only be run if you have a license for the GSC module.

Topics Covered

- Defining the Goal Seek and Control group
- Setting Variables and Goals
- Understanding Goal Seek and Control Output

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through the [Beginner - Heat Transfer in a Pipe - GSC](#) example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - NFPA Firewater System.fth***
- ***Metric - NFPA Firewater System - GSC.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the Model

Open the **Metric - NFPA Firewater System - Final.fth** file and save under a different name.

The workspace should look like Figure 1 below:

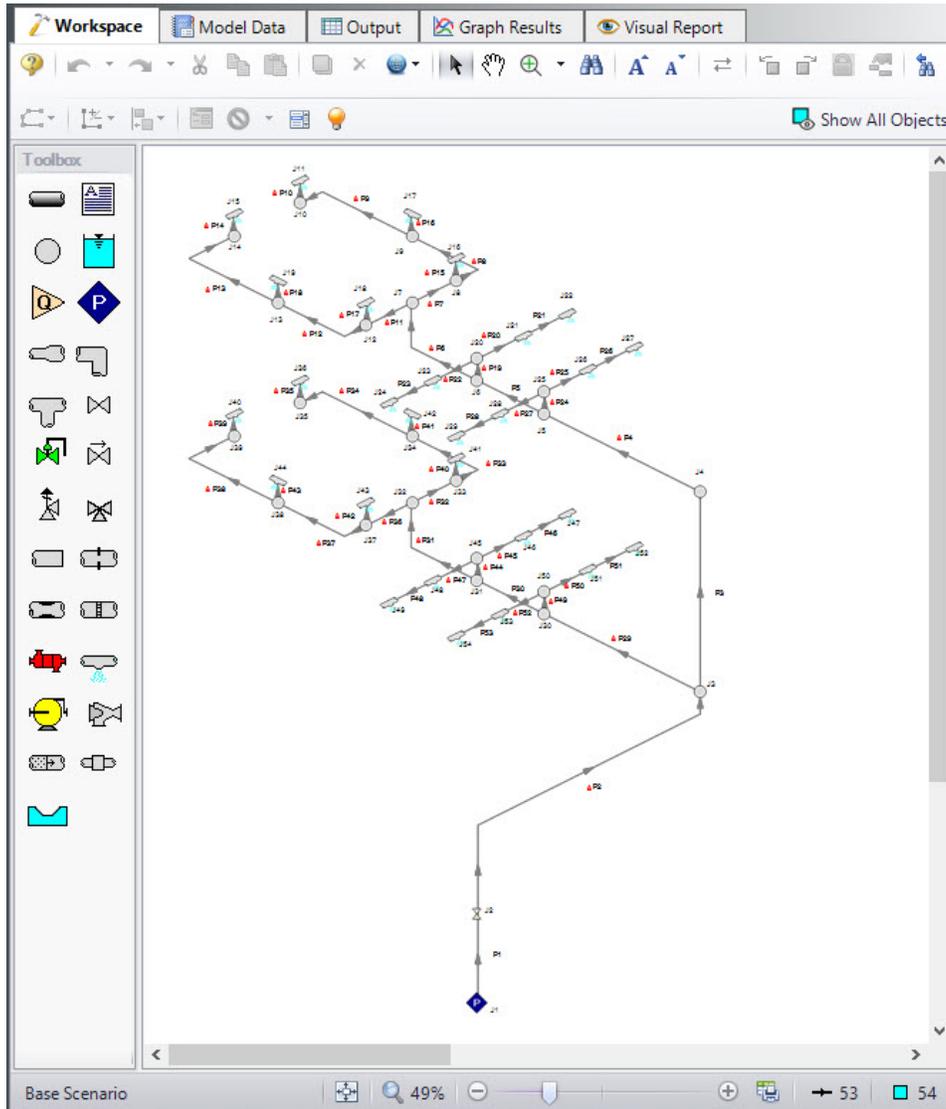


Figure 1: Workspace window for NFPA Firewater System - GSC Example

Step 3. Create the Spray Nozzles group

In this example, we will use the GSC module to determine the hydraulically most remote path.

To do this, you first need to create a group of all spray nozzles. Make this group by going to the Edit menu and selecting groups, and then clicking on Create. Name the group *Spray Nozzles* and under the

Junctions in Group section, click on Special, specify Selection Type as Junction Type, and choose Spray Discharge as the Junction Type. Click on Select Junctions and then press OK to confirm the selected junctions, then press OK to exit the Group Manager. 28 junctions should be selected.

Step 4. Define the Modules Group

Open Analysis Setup, and navigate to the Modules panel. Check the box next to Activate GSC. The Use option should automatically be selected, making GSC enabled for use.

Step 5. Define the Goal Seek and Control Group

Specify the variables and goals for the model.

Variables Panel

Next, ensure that GSC is both activated and enabled in the Modules panel of Analysis Setup. Then open the Variables panel. Click on New Variable and specify the Assigned Pressure J1 as the variable. This pressure boundary will be varied in order to achieve the goal, which in this case is a goal of at least 152 L/min through each sprinkler. See Figure 2 to see this variable specified.

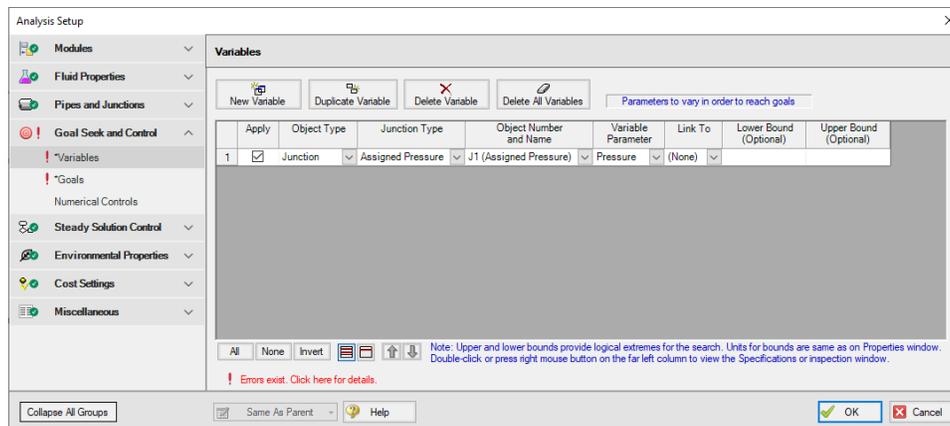


Figure 2: Assigned Pressure Variable on Goal Seek and Control Manager window

Goals Panel

Open the Goals panel and select New Goal. Specify the goal as shown in Figure 3.

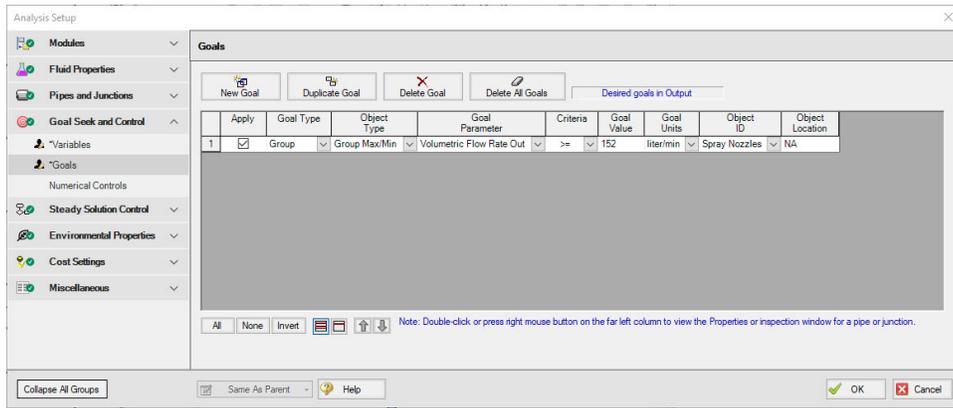


Figure 3: Minimum Flow Rate Group Goal on Goal Seek and Control Manager window

Step 6. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

You will notice there are numerous cautions about the pipe lengths being shorter than the elevation change. This model used pipe lengths and elevations based off a NFPA example. In the NFPA example, the length of the spray discharges raised the outlet height of the junction. Fathom treats junctions as single points in space, so to keep the inputs identical the pipe length had to be a little bit short.

Step 7. Examine the Output

The GSC module displays the value of the variable required in order to achieve the specified goal, as well as the object that was used for the group goal (in this case, this object is the spray discharge that receives the minimum amount of flow and, therefore, is the spray nozzle that contains the hydraulically most remote path). You can see that the pressure needed at J1 in order to ensure that all spray nozzles receive 152 L/min of flow is 4.406 barG, and that the hydraulically most remote spray nozzle is J15 (see Figure 4 and Figure 5).

Variable	Object Type	Object	Parameter	Value	Units
1	Assigned Pressure	J1	Pressure	4.406	barG

Figure 4: Calculated value for the GSC Variable in the Output

Goal	Goal Type	Object Type	Object ID	Object Location	Parameter	User Goal Value	Actual Goal Value	Units	Object Used for Group Goal
1	Group	Group Max/Min	Spray Nozzels	N/A	Volumetric Flow Rate Out	152	152.0	liter/min	J15 (Spray Discharge)

Figure 5: Calculated value for the GSC Goal when the GSC Variable is applied in the Output

Extended Time Simulation (XTS) Examples

The examples below utilize the Extended Time Simulation module.

Fathom XTS Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
Beginner - Filling a Tank	Beginner	Water	28	1	Learn the basics of the XTS module such as defining a transient junction and specifying the Transient Control group in Analysis Setup. See how to generate a Quick Graph from the Output window to see how a tank fills up over time.
Reservoir Balance	Beginner	Water	2	0	Learn how to use the XTS module to see the effects of a system with multiple finite open tanks.
Valve Closing	Beginner	Water	3	1	Learn how to use the XTS module to define an initiating transient event, such as a valve closing when a reservoir liquid height reaches a specified value.
Controlled Heat Exchanger Temperature	Beginner	Water	10	1	Learn how to use the XTS module to model a system with a pump that shuts off once the liquid level of a tank reaches a specified value.
Housing Project	Intermediate	Water	28	1	Learn how to use the XTS module to specify a Dual Event Cyclic transient initiation, such as a pump turning on or off depending on a reservoir liquid height.
Variable Demand	Intermediate	Water	19	2	Learn how to use the XTS module to define a Single Event Initiation of Transient for multiple junctions within a system.

Beginner - Filling a Tank - XTS

Summary

The objective of this example is to introduce the user to the Extended Time Simulation (XTS) module by modeling the filling of a tank. A liquid transfer system will be modeled. The pump will run for a specified time, and the change in fluid height in the delivery tank will be examined using the transient output data.

Note: This example can only be run if you have a license for the XTS module.

Topics Covered

- Specifying Transient Control
- Understanding Transient Output

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Beginner - Filling a Tank - XTS.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate XTS and select Transient to enable the XTS module for use. A

new group will appear in Analysis Setup titled Transient Control. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Time Simulation. From here you can quickly toggle between Steady Only mode (normal AFT Fathom) and Transient (XTS mode).

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = AFT Standard
 - b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
 - c. Temperature = 21 deg. C

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

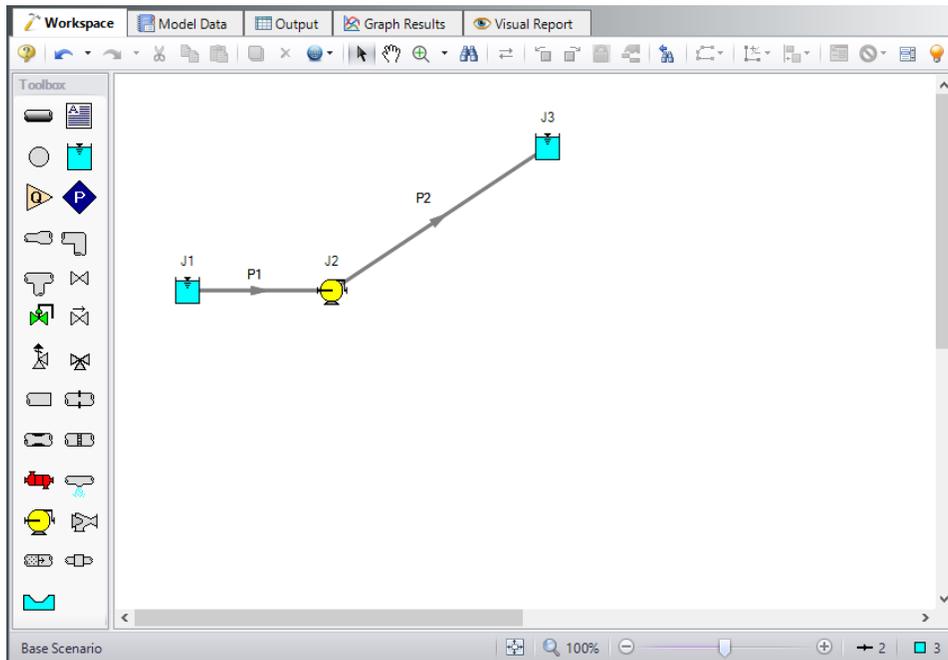


Figure 1: Workspace for XTS Filling a Tank example

The system is in place but now we need to enter the properties of the objects. Double-click each pipe and junction and enter the following properties. The required information is highlighted in blue.

Pipe Properties

1. Pipe Model tab
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 4 inch
 - d. Type = STD (schedule 40)
 - e. Friction Model Data Set = Standard
 - f. Lengths =

Pipe	Length (meters)
1	5
2	295

Junction Properties

1. J1 Reservoir
 - a. Name = Lower Reservoir
 - b. Tank Model = Infinite Reservoir
 - c. Liquid Surface Elevation = 3 meters
 - d. Liquid Surface Pressure = 0 barG (0 kPa (g))
 - e. Pipe Elevation = 0 meters

2. J2 Pump
 - a. Inlet Elevation = 0 meters
 - b. Pump Model = Centrifugal (Rotodynamic)
 - c. Analysis Type = Pump Curve
 - d. Enter Curve Data =

Volumetric	Head
m ³ /hr	meters
0	90
115	115
230	75

- e. Curve Fit Order = 2
3. J3 Reservoir
 - a. Name = Upper Reservoir
 - b. Tank Model = Finite Open Tank
 - c. Known Parameters Initially = Both Parameters (Liquid Surface Level & Surface Pressure)

- d. Cross-Sectional Area = Constant, 14 meters²
- e. Liquid Level Specification = Height from Bottom
- f. Liquid Height = 0 meters
- g. Liquid Surface Pressure = 0 barG (0 kPa (g))
- h. Tank Height = 9 meters
- i. Pipe Elevation = 60 meters
- j. Tank Bottom Elevation = 60 meters

To show the names of the reservoirs a new Workspace Layer will be created.

1. Create a new layer by selecting New Layer and then Blank Layer from the Workspace Layers tab of the Quick Access Panel.
2. Name the layer Reservoir Labels.
3. Edit the layer by selecting it and clicking the gear icon to bring up the Layer Settings window.
4. Open the Show/Hide Labels panel and uncheck Force Shown Labels to Match Shown Objects.
5. Toggle the visibility icon next to the reservoirs to on. This will cause their labels to be shown.
6. Navigate to the Junction Parameters panel and expand the Commonly Used Junction Parameters list.
7. Double-click Junction Name to add it to the list on the right-hand side. This will add the Junction Name to the label of the junctions that were defined in the Show/Hide Labels panel.
8. Close the Layer Settings window.

Step 5. Define the Transient Control Group

XTS module transient simulations are controlled through the Transient Control group in Analysis Setup. Access the Transient Control group by opening the Simulation Duration panel. When Fathom is set to perform a transient analysis, Transient Control becomes one of the items required to run the model.

The Simulation Duration panel in the Transient Control group is used to specify transient start and stop times, as well as the time step controls and solution parameters for finite tank liquid level calculations. Start times, stop times, and time steps can be specified in any units from seconds to years. From the Simulation Duration panel, users can specify the number of output points that are saved to the transient output file. This allows XTS to do transient calculations with a small time step while limiting the size of the transient data output file. Enter the following inputs in the Simulation Duration panel, as shown in Figure 2.

1. Start Time = 0 minutes
2. Stop Time = 30
3. Time Step = 1 minutes

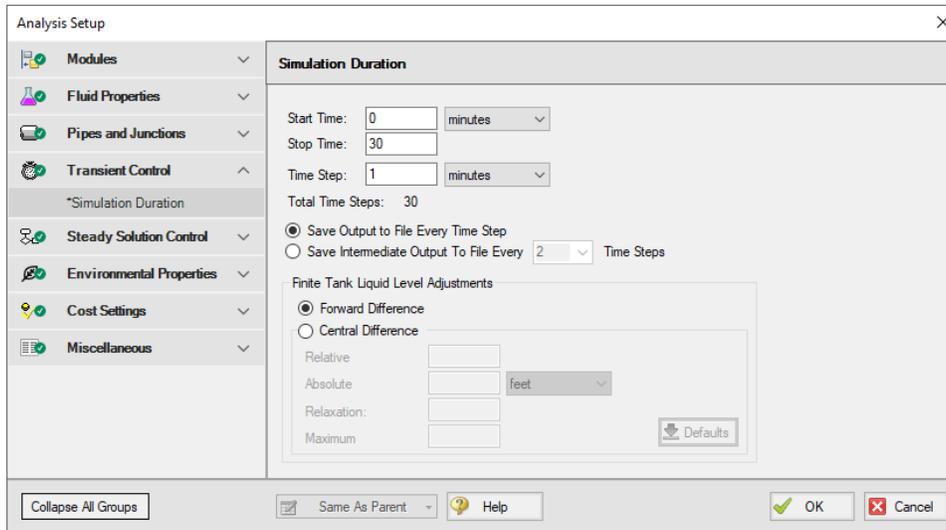


Figure 2: The Transient Control panel is used to control transient simulations

In addition to Transient Control, it is possible to define transient events for individual junctions. This allows the user to define events such as pump trips or start-ups, or valve closures. For this example, we will not specify any junction transients. The transient data for this example will be related to the fluid being pumped into the tank over the transient time period we have defined in Transient Control.

Step 6. Specify the Time Simulation Formatting

Open the Output Control window and select the Format & Action tab. Set the Time Simulation Formatting units to **minutes** and click OK to save and close the window.

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

After all of the pipes and junctions have been defined, and all of the transient data and transient control items have been specified, the transient simulation may be executed.

Step 7. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

When using the XTS module, the Solution Progress window displays the progress through the transient analysis. This information is displayed below the solution tolerance data, as shown in Figure 3:

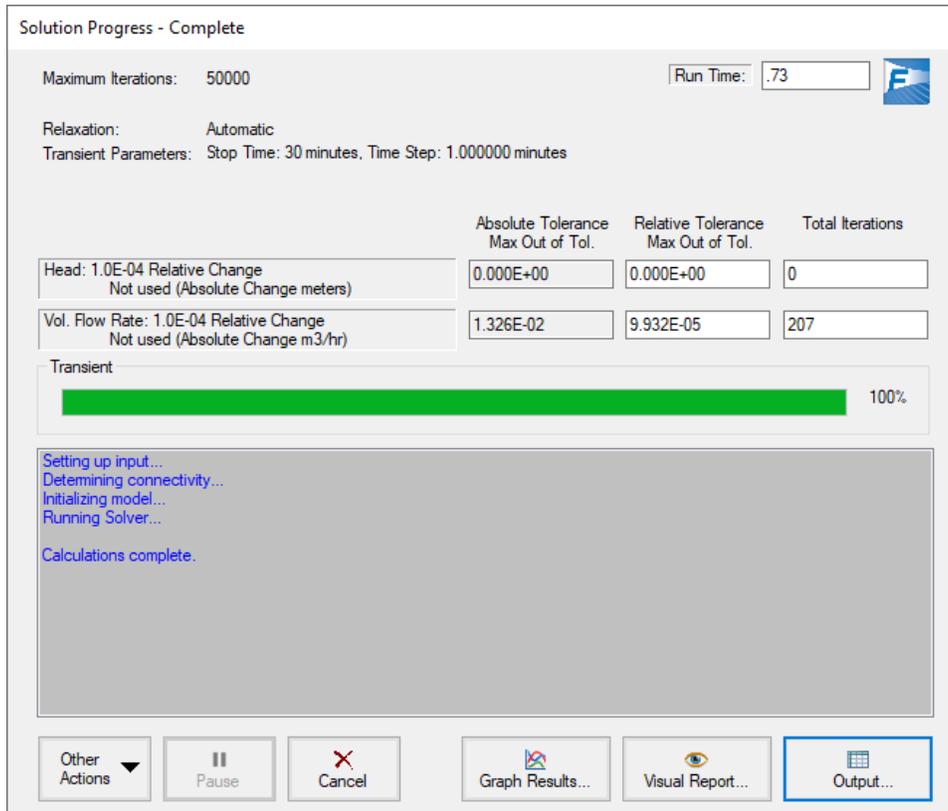


Figure 3: Solution Progress Window shows the convergence progress, as well as the transient solution progress when running transient cases

Step 8. Examine the Output

AFT Fathom displays transient output for all Pipes and Junctions in the Output Window, as shown in Figure 4.

Extended Time Simulation (XTS) Examples

The screenshot shows the 'Output' window with several summary tables. The 'Reservoir Transient' table shows data for two reservoirs at time 0. The 'Pipes Transient' table shows data for two pipes. The 'All Junctions Transient' table shows data for three junctions.

Tank Jct (Time)	Name	Type	Liq. Height (meters)	Liq. Elevation (meters)	Surface Pressure (bar)	Liquid Volume (meters ³)	Liquid Mass (kg)	Net Vol. Flow (m ³ /hr)	Net Mass Flow (kg/sec)
1 (0)	Lower Reservoir	Infinite	N/A	3.000	1.013	N/A	N/A	-136.6	-37.90
3 (0)	Upper Reservoir	Finite Open	0	60.000	1.013	0	0	136.6	37.90

Time: 0 Pipe	Name	Vol. Flow Rate (m ³ /hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)
1	Pipe	136.6	4.620	1.200	1.1100	0	0.00	0.09044	0.09044	0.000	0.9234	1.200	1.1100	1.307	1.217
2	Pipe	136.6	4.620	12.123	0.9067	0	60.00	11.21622	11.21622	5.876	54.5201	12.123	0.9067	12.229	1.013

Time: 0 Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m ³ /hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Lower Reservoir	1.013	1.307	1.013	1.307	136.6	37.90	0
2	Pump	1.110	12.123	1.217	12.229	136.6	37.90	0
3	Upper Reservoir	1.013	1.013	1.013	1.013	136.6	37.90	0

Figure 4: Transient output is displayed in the Output Window

Each of the summary tables in the General Output section has a companion transient summary tab which displays the summary data at each time step in the transient run. Each junction included in the summary is included in the transient summary. The transient summary data for each junction may be expanded or collapsed by clicking the + or – sign in beside the junction data list. The entire list may be expanded or collapsed by clicking the button in the top left-hand corner of the transient summary window. Figure 5 shows the Reservoir Transient summary tab, with all of the data sets collapsed. Figure 6 shows the Reservoir Transient summary tab, with the data for the J3 Reservoir expanded.

The screenshot shows the 'General Output' section with the 'Reservoir Transient' tab selected. It displays a table with transient summaries for two reservoirs.

Tank Jct (Time)	Name	Type	Liq. Height (meters)	Liq. Elevation (meters)	Surface Pressure (bar)	Liquid Volume (meters ³)	Liquid Mass (kg)	Net Vol. Flow (m ³ /hr)	Net Mass Flow (kg/sec)
1 (0)	Lower Reservoir	Infinite	N/A	3.000	1.013	N/A	N/A	-136.6	-37.90
3 (0)	Upper Reservoir	Finite Open	0	60.000	1.013	0	0	136.6	37.90

Figure 5: Transient Summaries are displayed in the General Output section

Extended Time Simulation (XTS) Examples

Reservoir Transient									
Tank Jct (Time)	Name	Type	Liq. Height (meters)	Liq. Elevation (meters)	Surface Pressure (bar)	Liquid Volume (meters ³)	Liquid Mass (kg)	Net Vol. Flow (m ³ /hr)	Net Mass Flow (kg/sec)
1 (0)	Lower Reservoir	Infinite	N/A	3.000	1.013	N/A	N/A	-136.6	-37.90
3 (0)	Upper Reservoir	Finite Open	0	60.000	1.013	0	0	136.6	37.90
3 (1)	Upper Reservoir	Finite Open	0.1626	60.163	1.013	2.277	2,274	136.5	37.87
3 (2)	Upper Reservoir	Finite Open	0.3252	60.325	1.013	4.552	4,546	136.4	37.83
3 (3)	Upper Reservoir	Finite Open	0.4875	60.487	1.013	6.825	6,816	136.2	37.78
3 (4)	Upper Reservoir	Finite Open	0.6496	60.650	1.013	9.094	9,083	136.0	37.73
3 (5)	Upper Reservoir	Finite Open	0.8115	60.812	1.013	11.36	11,347	135.8	37.69
3 (6)	Upper Reservoir	Finite Open	0.9732	60.973	1.013	13.63	13,608	135.7	37.64
3 (7)	Upper Reservoir	Finite Open	1.135	61.135	1.013	15.89	15,866	135.5	37.59
3 (8)	Upper Reservoir	Finite Open	1.296	61.296	1.013	18.14	18,122	135.3	37.54
3 (9)	Upper Reservoir	Finite Open	1.457	61.457	1.013	20.40	20,374	135.2	37.50
3 (10)	Upper Reservoir	Finite Open	1.618	61.618	1.013	22.65	22,624	135.0	37.45
3 (11)	Upper Reservoir	Finite Open	1.779	61.779	1.013	24.90	24,871	134.8	37.40
3 (12)	Upper Reservoir	Finite Open	1.939	61.939	1.013	27.15	27,115	134.6	37.35
3 (13)	Upper Reservoir	Finite Open	2.100	62.100	1.013	29.39	29,356	134.5	37.31
3 (14)	Upper Reservoir	Finite Open	2.260	62.260	1.013	31.64	31,595	134.3	37.26
3 (15)	Upper Reservoir	Finite Open	2.420	62.420	1.013	33.87	33,830	134.1	37.21
3 (16)	Upper Reservoir	Finite Open	2.579	62.579	1.013	36.11	36,063	134.0	37.17
3 (17)	Upper Reservoir	Finite Open	2.739	62.739	1.013	38.34	38,293	133.8	37.12
3 (18)	Upper Reservoir	Finite Open	2.898	62.898	1.013	40.57	40,520	133.6	37.07
3 (19)	Upper Reservoir	Finite Open	3.057	63.057	1.013	42.80	42,744	133.5	37.02
3 (20)	Upper Reservoir	Finite Open	3.216	63.216	1.013	45.02	44,966	133.3	36.98
3 (21)	Upper Reservoir	Finite Open	3.375	63.375	1.013	47.24	47,184	133.1	36.93
3 (22)	Upper Reservoir	Finite Open	3.533	63.533	1.013	49.46	49,400	132.9	36.88
3 (23)	Upper Reservoir	Finite Open	3.691	63.691	1.013	51.68	51,613	132.8	36.83
3 (24)	Upper Reservoir	Finite Open	3.849	63.849	1.013	53.89	53,822	132.6	36.78
3 (25)	Upper Reservoir	Finite Open	4.007	64.007	1.013	56.10	56,029	132.4	36.73
3 (26)	Upper Reservoir	Finite Open	4.165	64.165	1.013	58.31	58,234	132.2	36.69
3 (27)	Upper Reservoir	Finite Open	4.322	64.322	1.013	60.51	60,435	132.1	36.64
3 (28)	Upper Reservoir	Finite Open	4.480	64.480	1.013	62.71	62,633	131.9	36.59
3 (29)	Upper Reservoir	Finite Open	4.637	64.637	1.013	64.91	64,829	131.7	36.54
3 (30)	Upper Reservoir	Finite Open	4.793	64.793	1.013	67.11	67,021	131.6	36.50

Figure 6: Transient Summary data can be expanded to show the junction data for every Time Step

The Quick Graph feature can be used to quickly plot transient data for quick examination. To use the Quick Graph feature, place the mouse cursor over the column of data you wish to examine. Then, right click with the mouse, and select Quick Graph from the list. Figure 7 illustrates how to do this for J3 Reservoir Liquid Height vs. Time. Figure 8 shows the resulting graph. The graph illustrates how the liquid level in the Reservoir rises with time, as would be expected. The Graph Results window may also be used to create a full range of graphs related to the XTS module transient output.

Extended Time Simulation (XTS) Examples

Tank Jct (Time)	Name	Type	Liq. Height (meters)	Liq. Elevation (meters)	Surface Pressure (bar)	Liquid Volume (meters ³)	Liquid Mass (kg)	Net Vol. Flow (m ³ /hr)	Net Mass Flow (kg/sec)
1 (0)	Lower Reservoir	Infinite	N/A	3.000	1.013	N/A	N/A	-136.6	-37.90
3 (0)	Upper Reservoir	Finite Open	0	60.000	1.013	0	0	136.6	37.90
3 (1)	Upper Reservoir	Finite Open	0.1						37.87
3 (2)	Upper Reservoir	Finite Open	0.3						37.83
3 (3)	Upper Reservoir	Finite Open	0.4						37.78
3 (4)	Upper Reservoir	Finite Open	0.6						37.73
3 (5)	Upper Reservoir	Finite Open	0.8						37.69
3 (6)	Upper Reservoir	Finite Open	0.9						37.64
3 (7)	Upper Reservoir	Finite Open	1						37.59
3 (8)	Upper Reservoir	Finite Open	1.296	61.296	1.013	18.14	18,122	135.3	37.54
3 (9)	Upper Reservoir	Finite Open	1.457	61.457	1.013	20.40	20,374	135.2	37.50
3 (10)	Upper Reservoir	Finite Open	1.618	61.618	1.013	22.65	22,624	135.0	37.45

Figure 7: Use Quick Graph to easily plot J3 Reservoir liquid height

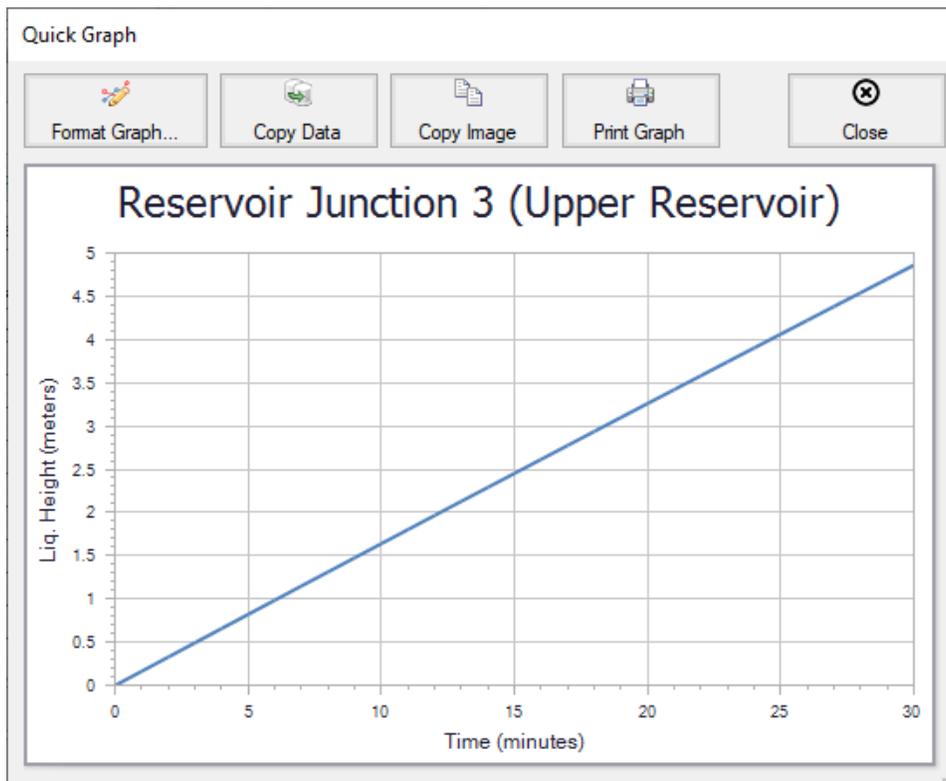


Figure 8: Quick Graph Plot of J3 Reservoir liquid height

Similar to the summary transient data, the transient tabs in the Pipe and Junctions sections of the Output Window are used to display the transient data for each pipe and junction at every time step, as shown in Figure 9. This transient data can be expanded or collapsed in the same manner as in the General Output transient summaries.

Extended Time Simulation (XTS) Examples

Pipes		Transient													
Pipe (Time)	Name	Vol. Flow Rate (m3/hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)
1 (0)	Pipe	136.6	4.620	1.200	1.1100	0	0.00	0.09044	0.09044	0.000	0.9234	1.200	1.1100	1.307	1.217
2 (0)	Pipe	136.6	4.620	12.123	0.9067	0	60.00	11.21622	11.21622	5.876	54.5201	12.123	0.9067	12.229	1.013
2 (1)	Pipe	136.5	4.617	12.125	0.9227	0	60.00	11.20182	11.20182	5.876	54.3731	12.125	0.9227	12.231	1.029
2 (2)	Pipe	136.4	4.612	12.128	0.9389	0	60.00	11.18899	11.18899	5.876	54.2420	12.128	0.9389	12.234	1.045
2 (3)	Pipe	136.2	4.606	12.131	0.9551	0	60.00	11.17613	11.17613	5.876	54.1108	12.131	0.9551	12.237	1.061
2 (4)	Pipe	136.0	4.600	12.134	0.9712	0	60.00	11.16328	11.16328	5.876	53.9796	12.134	0.9712	12.240	1.077
2 (5)	Pipe	135.8	4.594	12.138	0.9873	0	60.00	11.15044	11.15044	5.876	53.8484	12.138	0.9873	12.243	1.093
2 (6)	Pipe	135.7	4.589	12.141	1.003	0	60.00	11.13760	11.13760	5.876	53.7173	12.141	1.003	12.246	1.109
2 (7)	Pipe	135.5	4.583	12.144	1.020	0	60.00	11.12477	11.12477	5.876	53.5863	12.144	1.020	12.249	1.124
2 (8)	Pipe	135.3	4.577	12.148	1.036	0	60.00	11.11195	11.11195	5.876	53.4554	12.148	1.036	12.252	1.140
2 (9)	Pipe	135.2	4.571	12.151	1.052	0	60.00	11.09914	11.09914	5.876	53.3246	12.151	1.052	12.255	1.156
2 (10)	Pipe	135.0	4.566	12.154	1.068	0	60.00	11.08633	11.08633	5.876	53.1938	12.154	1.068	12.258	1.172

All Junctions		Transient										
Jct (Time)	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)				
1 (0)	Lower Reservoir	1.013	1.307	1.013	1.307	136.6	37.90	0				
2 (0)	Pump	1.110	12.123	1.217	12.229	136.6	37.90	0				
3 (0)	Upper Reservoir	1.013	1.013	1.013	1.013	136.6	37.90	0				
3 (1)	Upper Reservoir	1.013	1.029	1.013	1.029	136.5	37.87	0				
3 (2)	Upper Reservoir	1.013	1.045	1.013	1.045	136.4	37.83	0				
3 (3)	Upper Reservoir	1.013	1.061	1.013	1.061	136.2	37.78	0				
3 (4)	Upper Reservoir	1.013	1.077	1.013	1.077	136.0	37.73	0				
3 (5)	Upper Reservoir	1.013	1.093	1.013	1.093	135.8	37.69	0				
3 (6)	Upper Reservoir	1.013	1.109	1.013	1.109	135.7	37.64	0				
3 (7)	Upper Reservoir	1.013	1.124	1.013	1.124	135.5	37.59	0				

Figure 9: Output Window showing the Pipe and Junction Transient tabs

Figure 10 shows the output data for the pipes and junctions, found on the Pipes and All Junctions tabs, at the initial time step, Time = 0 minutes. The output for all of the pipes and junctions can be displayed at any time step by using the slider bar located at the bottom of the Output window. Figure 11 shows the pipe and junction data at Time = 5 minutes.

Pipes		Transient													
Time: 0 Pipe	Name	Vol. Flow Rate (m3/hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)
1	Pipe	136.6	4.620	1.200	1.1100	0	0.00	0.09044	0.09044	0.000	0.9234	1.200	1.1100	1.307	1.217
2	Pipe	136.6	4.620	12.123	0.9067	0	60.00	11.21622	11.21622	5.876	54.5201	12.123	0.9067	12.229	1.013

All Junctions		Transient										
Time: 0 Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)				
1	Lower Reservoir	1.013	1.307	1.013	1.307	136.6	37.90	0				
2	Pump	1.110	12.123	1.217	12.229	136.6	37.90	0				
3	Upper Reservoir	1.013	1.013	1.013	1.013	136.6	37.90	0				

Time: 0 minutes

Figure 10: Pipe and Junction transient data at Time = 0 minutes

Extended Time Simulation (XTS) Examples

Time: 5		Pipes													
Pipe	Name	Vol. Flow Rate (m3/hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)
1	Pipe	135.8	4.594	1.202	1.1122	0	0.00	0.08946	0.08946	0.000	0.9134	1.202	1.1122	1.307	1.218
2	Pipe	135.8	4.594	12.138	0.9873	0	60.00	11.15044	11.15044	5.876	53.8484	12.138	0.9873	12.243	1.093

Time: 5		All Junctions							
Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)	
1	Lower Reservoir	1.013	1.307	1.013	1.307	135.8	37.69	0	
2	Pump	1.112	12.138	1.218	12.243	135.8	37.69	0	
3	Upper Reservoir	1.013	1.093	1.013	1.093	135.8	37.69	0	

Time: 5 minutes

Figure 11: Pipe and Junction transient data at Time = 5 minutes

Step 9. Create a New Scenario

Create a new scenario called Variable Speed Pump.

Open the Pump Properties window for J2. On the Variable Speed tab, set the pump to Controlled Pump (Variable Speed) and select Volumetric Flow Rate with a Control Setpoint of 100 m3/hr.

Rerun the model.

Step 10. Examine the Output

Figure 12 shows a Quick Graph plot of the Pump Speed vs. Time for the controlled flow pump. Note how the pump speed increases over time to maintain the controlled flow. This is because the hydrostatic pressure downstream of the pump is increasing because the tank is slowly filling over time.

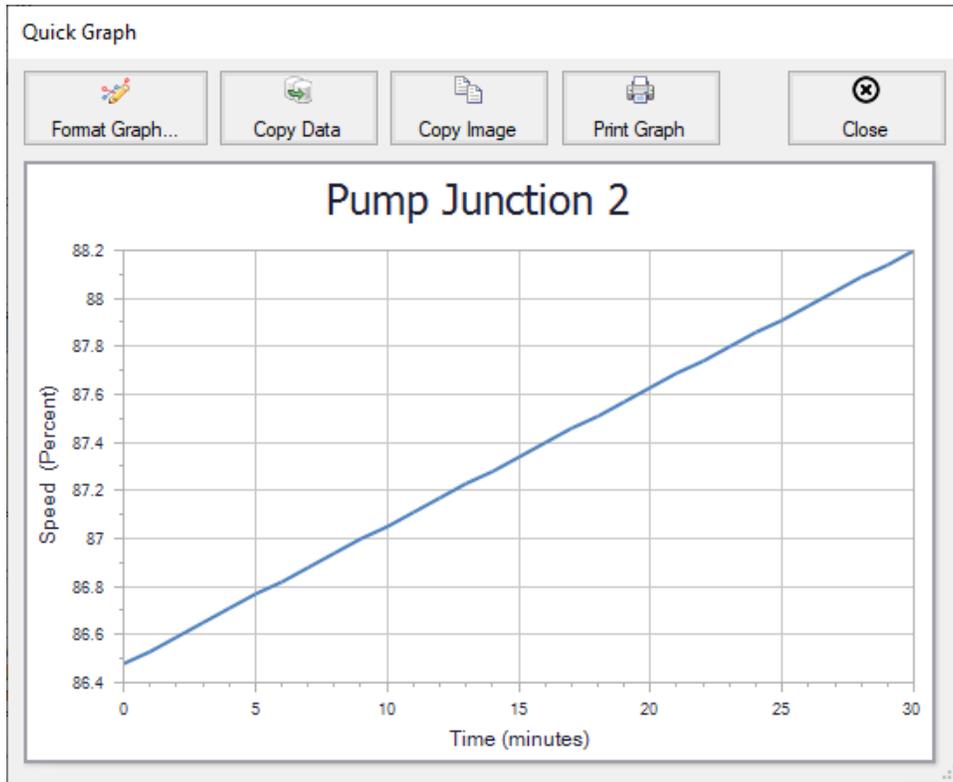


Figure 12: Quick Graph plot of Controlled Pump Speed vs. Time

Reservoir Balance - XTS

Summary

The objective of this example is to demonstrate how the interaction of different hydrostatic pressures between can be analyzed using the XTS module. This example will use three finite reservoirs connected in series, with different initial liquid elevations.

Note: This example can only be run if you have a license for the XTS module.

Topics Covered

- Using Finite Reservoirs Junctions

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the [Beginner - Filling a Tank - XTS](#) example, and is familiar with the basics of XTS transient analysis.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Reservoir Balance - XTS.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate XTS and select Transient to enable the XTS module for use. A new group will appear in Analysis Setup titled Transient Control. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Time Simulation. From here you can quickly toggle between Steady Only mode (normal AFT Fathom) and Transient (XTS mode).

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = AFT Standard
 - b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
 - c. Temperature = 21 deg. C

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

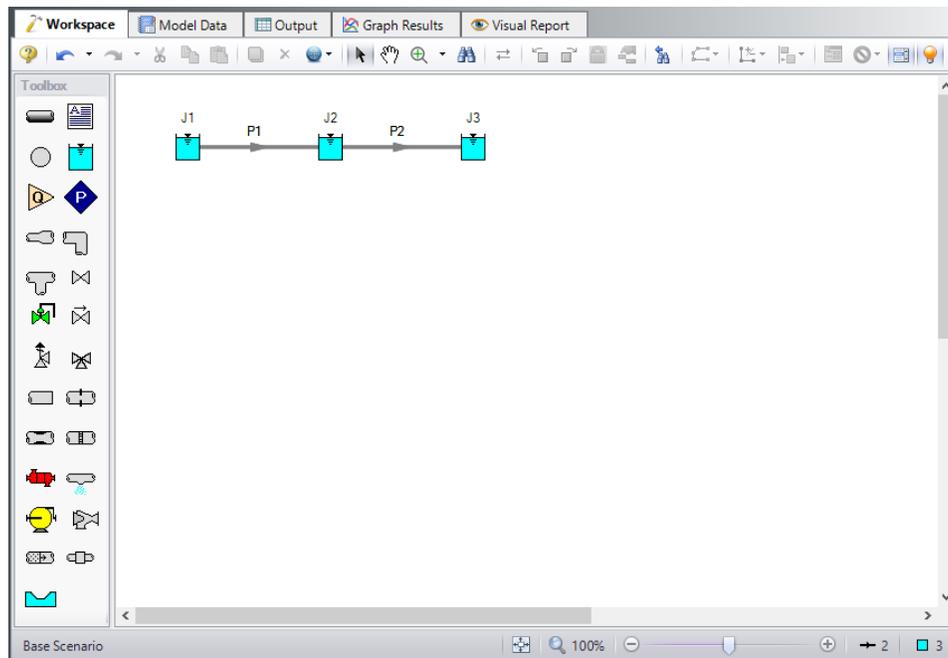


Figure 1: Workspace for XTS Reservoir Balance example

Pipe Properties

Enter the following for both pipes P1 and P2.

1. Pipe Model tab
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 4 inch
 - d. Type = STD (schedule 40)
 - e. Friction Model Data Set = Standard
 - f. Length = 75 meters

Junction Properties

Figure 2 shows the Reservoir Properties window for junction J1. In the Fathom XTS module, reservoir junctions can have finite dimensions. The reservoirs can also be open, or closed. If the tank is closed, the user must specify the gas thermodynamics inside the closed tank. Users may also specify different initial known parameters for the reservoir. If only the Liquid Surface level is known, Fathom will calculate the Surface Pressure. Alternatively, if only the Surface Pressure is known, Fathom will calculate the Liquid Height. When using a reservoir as a finite tank, users must also specify the tank height, the initial liquid height in the tank, as well as the tank cross sectional area.

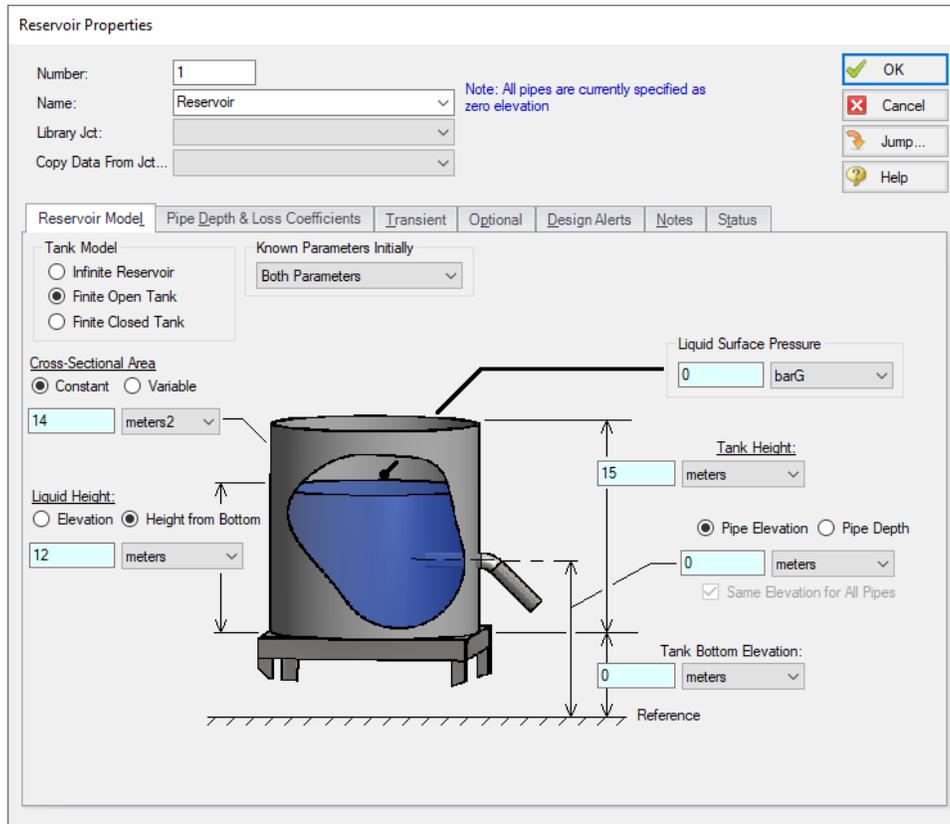


Figure 2: Reservoir Properties Window in Fathom XTS

Next, define the reservoir junctions.

1. J1 Reservoir
 - a. Tank Model = Finite Open Tank
 - b. Known Parameters Initially = Both Parameters (i.e. Liquid Surface Level & Surface Pressure)
 - c. Cross-Sectional Area = Constant, 14 meters²
 - d. Liquid Level Specification = Height from Bottom
 - e. Liquid Height = 12 meters
 - f. Liquid Surface Pressure = 0 barG (kPa (g))
 - g. Tank Height = 15 meters
 - h. Pipe Elevation = 0 meters
 - i. Tank Bottom Elevation = 0 meters
2. J2 Reservoir
 - a. Tank Model = Finite Open Tank
 - b. Known Parameters Initially = Both Parameters (i.e. Liquid Surface Level & Surface Pressure)
 - c. Cross-Sectional Area = Constant, 14 meters²

- d. Liquid Level Specification = Height from Bottom
 - e. Liquid Height = 0 meters
 - f. Liquid Surface Pressure = 0 barG (kPa (g))
 - g. Tank Height = 15 meters
 - h. Pipe Elevation = 0 meters
 - i. Tank Bottom Elevation = 0 meters
3. J3 Reservoir
- a. Tank Model = Finite Open Tank
 - b. Known Parameters Initially = Both Parameters (i.e. Liquid Surface Level & Surface Pressure)
 - c. Cross-Sectional Area = Constant, 14 meters²
 - d. Liquid Level Specification = Height from Bottom
 - e. Liquid Height = 3 meters
 - f. Liquid Surface Pressure = 0 barG (kPa (g))
 - g. Tank Height = 15 meters
 - h. Pipe Elevation = 0 meters
 - i. Tank Bottom Elevation = 0 meters

Step 5. Define the Transient Control Group

Open Analysis Setup again and navigate to the Simulation Duration panel, part of the Transient Control group.

For this example, with a Start Time of 0 minutes, set the Stop Time to 120 minutes with a time step of 1 minute. The output will be saved to the output file for every time step.

Step 6. Specify the Time Simulation Formatting

Open the Output Control window and select the Format & Action tab. Set the Time Simulation Formatting units to **minutes** and click OK to save and close the window.

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 7. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 8. Examine the Output

For this model, the output will be easiest to view as a graph, so select the Graph Results tab. Use the Graph Results window to plot the liquid height in each of the three reservoirs on the same plot. Figure 3 shows the Graph Control tab of the Quick Access Panel for simultaneously plotting the transient liquid height in all three reservoirs. After selecting the Transient graphing tab, select Reservoir from the Pipes or Junctions drop-down menu.

Figure 4 show the resulting graph. Note that, as one would expect because the tanks are the same sizes and at the same elevations, the liquid heights between the tanks eventually reach the same equilibrium value.

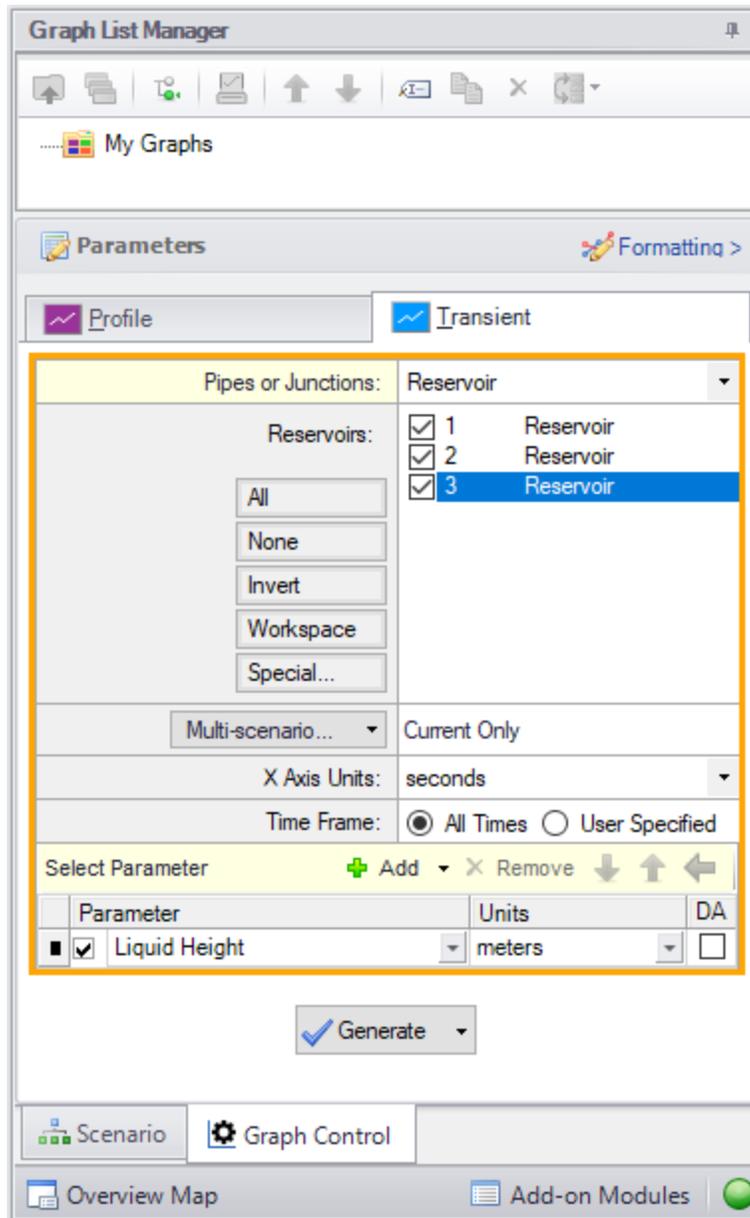


Figure 3: The transient liquid height in all three reservoirs can be selected in the Select Graph Data window

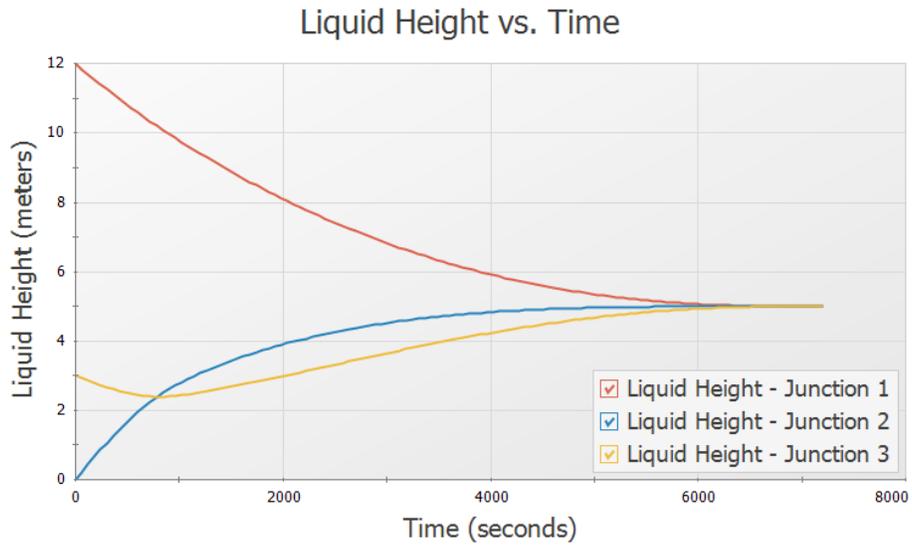


Figure 4: Reservoir liquid heights can be plotted using the Graph Results window

Valve Closing - XTS

Summary

The objective of this example is to demonstrate how to set up an event based transient in the XTS module for a valve junction. A valve is set to close when the liquid height in discharge tank reaches a specified level.

Note: This example can only be run if you have a license for the XTS module.

Topics Covered

- Defining Junction Transients
- Understanding Transient Output

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the [Beginner - Filling a Tank - XTS](#) example, and is familiar with the basics of XTS transient analysis.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Valve Closing - XTS.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate XTS and select Transient to enable the XTS module for use. A new group will appear in Analysis Setup titled Transient Control. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Time Simulation. From here you can quickly toggle between Steady Only mode (normal AFT Fathom) and Transient (XTS mode).

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = AFT Standard
 - b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
 - c. Temperature = 21 deg. C

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

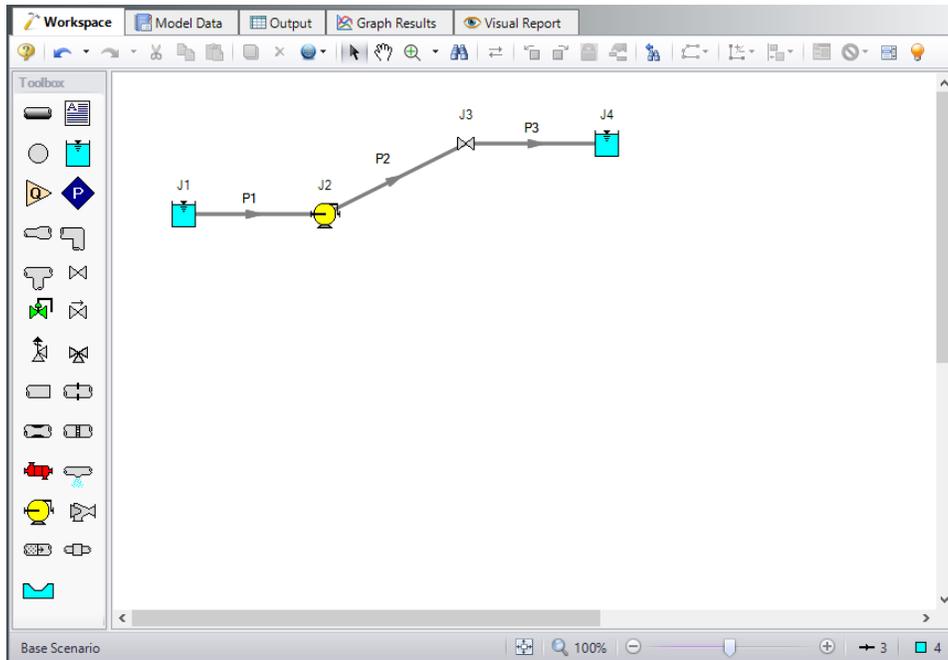


Figure 1: Workspace for XTS Valve Closing example

The system is in place but now we need to enter the properties of the objects. Double-click each pipe and junction and enter the following properties. The required information is highlighted in blue.

Pipe Properties

1. Pipe Model tab
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 4 inch
 - d. Type = STD (schedule 40)
 - e. Friction Model Data Set = Standard
 - f. Lengths =

Pipe	Length (meters)
1	6
2	300
3	6

Junction Properties

1. J1 Reservoir
 - a. Name = Lower Reservoir
 - b. Tank Model = Infinite Reservoir
 - c. Liquid Surface Elevation = 3 meters
 - d. Liquid Surface pressure = 0 barG (0 kPa (g))
 - e. Pipe Depth = 3 meters

2. J2 Pump
 - a. Inlet Elevation = 0 meters
 - b. Pump Model = Centrifugal (Rotodynamic)
 - c. Analysis Type = Pump Curve
 - d. Enter Curve Data =

Volumetric	Head
m3/hr	meters
0	90
115	115
230	75

- e. Curve Fit Order = 2

1. J3 Valve
 - a. Inlet Elevation = 60 meters
 - b. Valve Data Source = User Specified
 - c. Loss Model = Cv
 - d. Loss Source = Fixed Cv
 - e. Cv = 100

3. J4 Reservoir
 - a. Name = Upper Reservoir
 - b. Tank Model = Finite Open Tank
 - c. Known Parameters Initially = Both Parameters
 - d. Cross-Sectional Area = Constant, 14 meters²
 - e. Liquid Height Specification = Height from Bottom
 - f. Liquid Height = 0 meters
 - g. Liquid Surface Pressure = 0 barG (0 kPa (g))
 - h. Tank Height = 9 meters
 - i. Pipe Elevation = 60 meters
 - j. Tank Bottom Elevation = 60 meters

Step 5. Define the Transient Control Group

Open Analysis Setup again and navigate to the Simulation Duration panel, part of the Transient Control group.

For this example, with a Start Time of 0 minutes, set the Stop Time to 60 minutes with a time step of 1 minute. The output will be saved to the output file for every time step.

Step 6. Specify the Time Simulation Formatting

Open the Output Control window and select the Format & Action tab. Set the Time Simulation Formatting units to **minutes** and click OK to save and close the window.

Step 7. Enter the Valve Transient Data

For this example, we will set an event-based transient on the valve junction such that when the liquid elevation in the reservoir reaches a height of 5 meters, the valve will close over a period of 30 seconds. Junction transient events are defined on the Transient Data tabs in the Junction Properties windows. Open the J3 Valve Properties window, and select the Transient tab.

Transients for junctions can be defined as either time based or event based transients. A time based transient starts at a specified absolute time. Event based transients are initiated when a defined criteria, such as when a pressure in a particular pipe is reached or exceeded. There are three types of event transients for junctions. Single event transients are events that occur only once in a simulation. Dual cyclic events are two transient events that repeat one after the other as many times as the event triggers occur during a simulation (such as a valve closing then reopening). Dual sequential transient events are two transients events which occur one after the other, without repeating.

For this example, we will define a single transient event. The event will occur when the liquid height in the upper reservoir reaches 5 meters. When the liquid height reaches 5 meters, the valve will close over a period of 30 seconds.

Figure 2 shows the Transient Data tab for the Valve Properties window. The transient event is defined in the Initiation of Transient area. This is where the event that will trigger the transient event is defined. The profile of what occurs when the transient is initiated is defined in the Transient Data area. For this example, the transient data consists of the valve closing profile. The valve Cv profile is specified over the time period that the valve is to close. In this example, the Cv profile is linear over the 30-second closing time. Enter the transient data as shown below and in Figure 2, then select OK.

1. Transient tab
 - a. Transient Special Condition = None
 - b. Initiation of Transient = Single Event
 - c. Event Type = Reservoir Liquid Height
 - d. Condition = Greater Than
 - e. Value = 5 meters
 - f. Junction = 4 (Upper Reservoir)
 - g. Transient Data = Absolute Values
 - h. Data Points =

Time (seconds)	Cv
0	100

Time (seconds)	Cv
30	0
3600	0

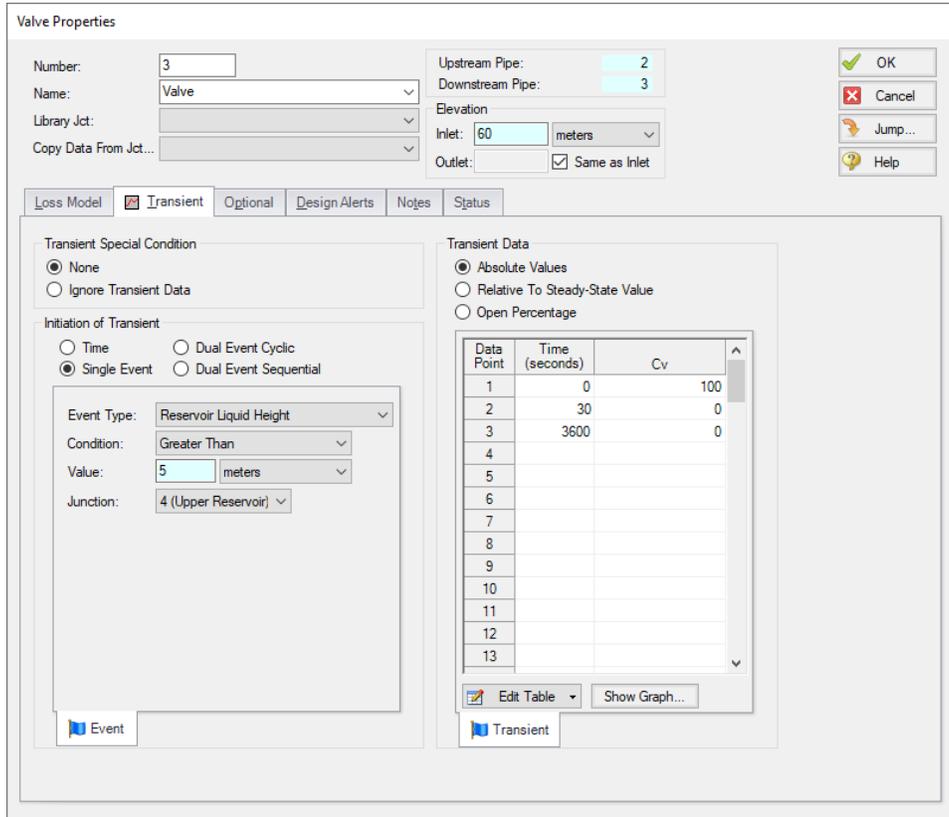


Figure 2: Valve closing Event Transient definitions are made on the Transient tab in the Valve Properties window

Fathom will display a T next to the junction label on the workspace when a transient has been defined for a junction.

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 8. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 9. Examine the Output

When junction event transients occur, these events are recorded and displayed in the general output section of the Output Window. Open the Output Control and navigate to Format & Action. Change the Time Simulation Formatting to minutes. The events are displayed on two different tabs. The first tab sorts the events by junction number, and the second tab sorts the events by time, in the order in which they occur. The tabs are shown in Figure 3:

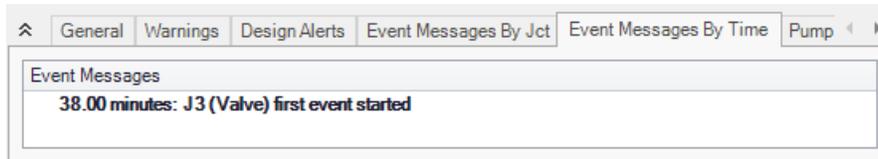


Figure 3: Event Transient Tabs in are shown in the General Output section of the Output Window

A Quick Graph plot of the liquid height in the reservoir, shown in Figure 4, shows that the final liquid height in the Upper Reservoir is about 5.2 meters. This is because the valve closes over a 30-second period. The valve starts to close when the liquid reaches 5 meters. During the 30-second period that the valve is closing, liquid continues to flow into the tank. Thus, there is some *overshoot* which results in approximately 20 additional centimeters of fluid being added to the tank. This can also be seen by looking at the liquid height at the final time step shown on the Reservoir Transient tab.

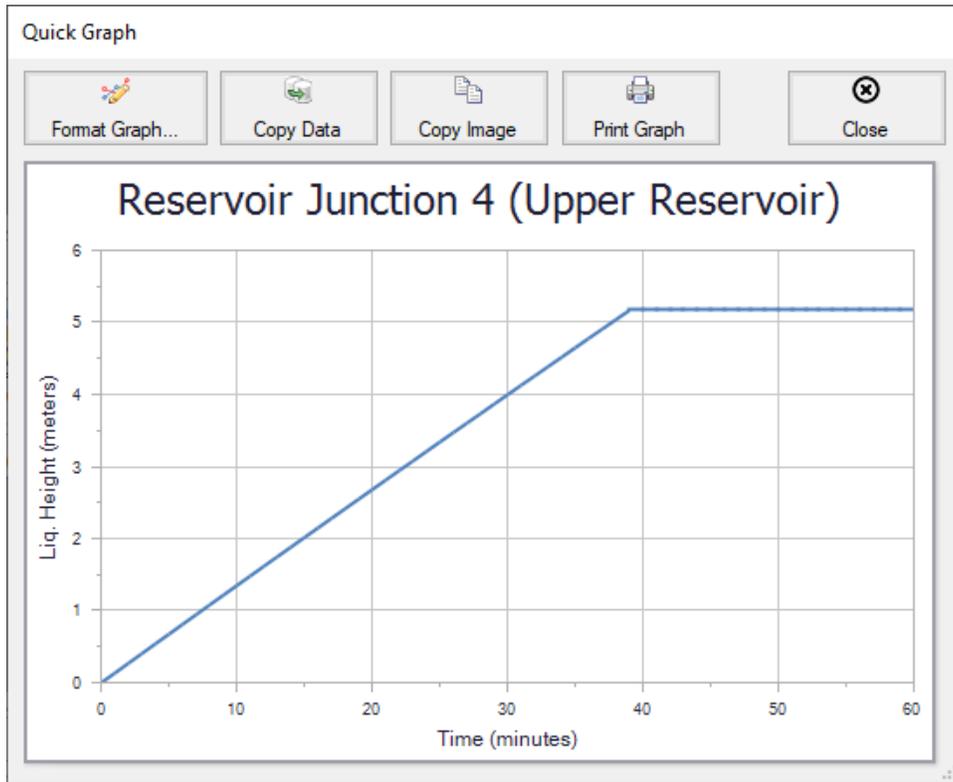


Figure 4: A Quick Graph plot can be used to plot reservoir liquid height showing overshoot due to valve closure time

Controlled Heat Exchanger Temperature - XTS

Summary

Use the Extended Time Simulation module to determine the length of time required to fill the discharge reservoir, and to turn off the pump when the tank is full.

The discharge reservoir is a finite size tank, 6 meters high, with a cross sectional area of 7.5 meters². The pump is to shut off when the tank is filled. How long does it take to fill the tank? What happens after the pump is turned off?

Note: This example can only be run if you have a license for the XTS module.

Topics Covered

- Defining Transient Control
- Defining system transients

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the [Beginner - Filling a Tank - XTS](#) example, and is familiar with the basics of XTS transient analysis.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Controlled Heat Exchanger Temperature.fth***
- ***Metric - Controlled Heat Exchanger Temperature - XTS.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the Model

Open the **Metric - Controlled Heat Exchanger Temperature.fth** example file and save it as a new file. The workspace should look like Figure 1 below.

Important: The XTS and GSC modules and cost calculations can be used in the same run to simulate control functions over time, but we are not going to do this here.

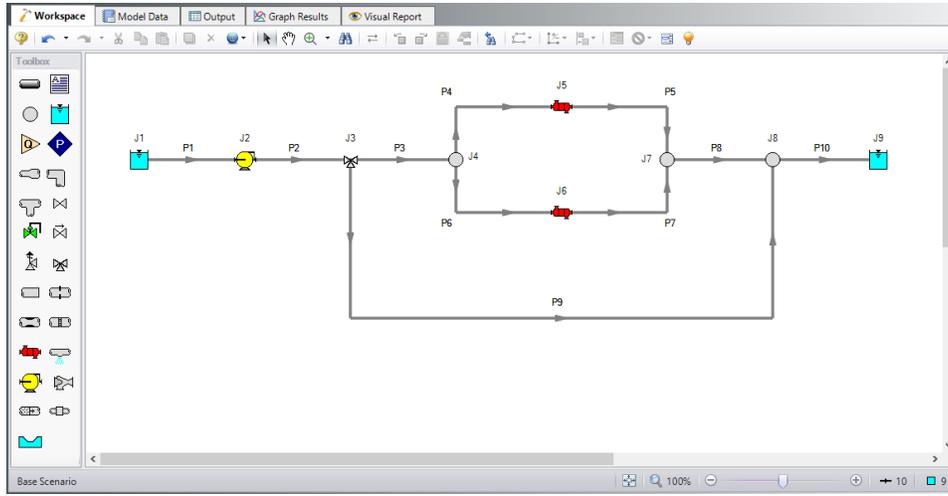


Figure 1: Workspace for XTS Controlled Heat Exchanger Temperature example

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate XTS and select Transient to enable the XTS module for use. A new group will appear in Analysis Setup titled Transient Control. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Time Simulation. From here you can quickly toggle between Steady Only mode (normal AFT Fathom) and Transient (XTS mode).

Note: The XTS module does not perform heat transfer calculations. You will be prompted to disable heat transfer calculations in order to proceed with modeling a transient. Click Yes.

Step 4. Define Pipes and Junctions group

Most of the input is already entered. The only exception is for Reservoir J9. Enter the following junction properties.

1. J3 Three Way Valve
 - a. Library Jct = (None)
 - b. Actual Percent Open = 79 %
 - i. This value comes from the Actual Percent Open value calculated from the GSC example
2. J9 Reservoir
 - a. Name = Discharge Reservoir
 - b. Tank Model = Finite Open Tank
 - c. Known Parameters Initially = Liquid Surface Level & Surface Pressure
 - d. Cross-Sectional Area = Constant, 7.5 meters²
 - e. Liquid Level Specification = Height from Bottom
 - f. Liquid Height = 0 meters
 - g. Liquid Surface Pressure = 0.3 barG (30 kPa (g))
 - h. Tank Height = 6 meters
 - i. Pipe Elevation = 6 meters
 - j. Tank Bottom Elevation = 6 meters

Step 5. Setup the Transient Data

For this example, we will set an event-based transient on the pump so the pump will turn off when the liquid level in the discharge tank reaches 6 meters.

You will define a single transient event. The event will occur when the liquid height in the reservoir reaches 6 meters. When the liquid height reaches 6 meters, the pump speed will drop from 100% to 0% in 10 seconds.

Open the Pump Properties window, and enter the following information. Then the Transient tab should appear as shown in Figure 2.

1. Library Jct = (None)
2. Transient tab
 - a. Transient Type = Speed vs. Time
 - b. Transient Special Condition = None
 - c. Initiation of Transient = Single Event
 - d. Event Type = Reservoir Liquid Height
 - e. Condition = Greater Than or Equal To
 - f. Value = 6 meters
 - g. Junction = 9 (Discharge Reservoir)
 - h. Transient Data = Absolute Values
 - i. Data Points =

Time (seconds)	Speed (Percent)
0	100
10	0

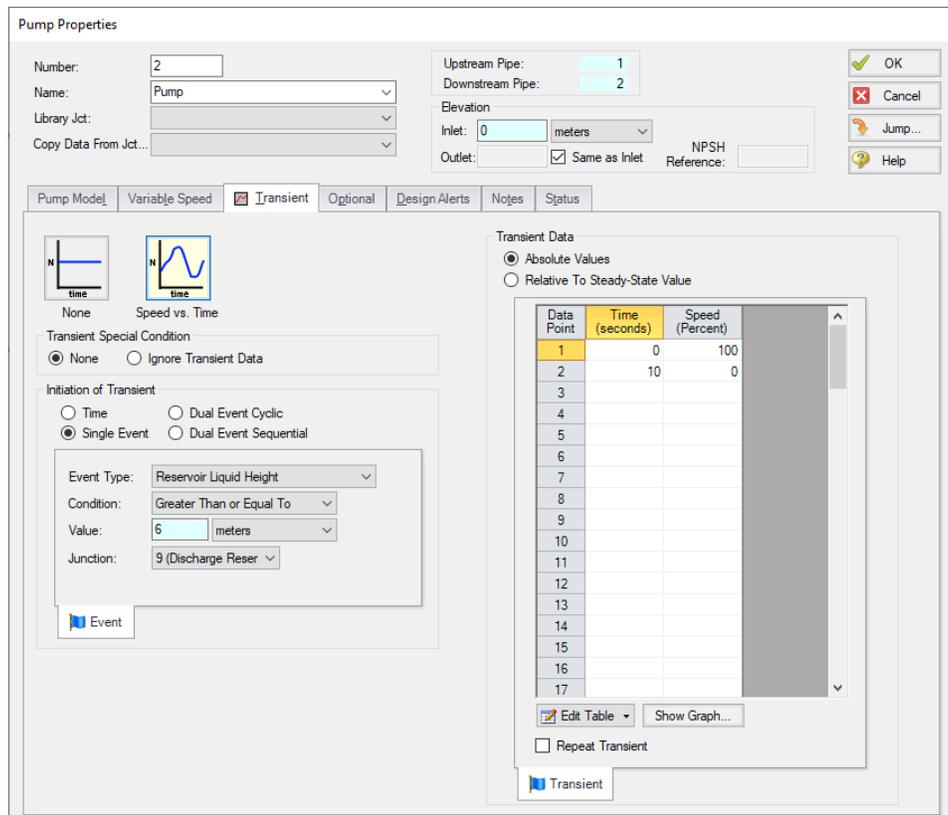


Figure 2: Junction transients are specified on the Transient tabs on the junction Properties windows

➤ Turn on **Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 6. Define the Transient Control Group

Open Analysis Setup again and navigate to the Simulation Duration panel, part of the Transient Control group.

For this example, with a Start Time of **0 minutes**, set the Stop Time to **45 minutes** with a time step of **1 minute**. The output will be saved to the output file for every time step.

Step 7. Specify the Time Simulation Formatting

Open the Output Control window and select the Format & Action tab. Set the Time Simulation Formatting units to **minutes** and click OK to save and close the window.

Step 8. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 9. Examine the Output

Open the Output Control and navigate to Format & Action. Change the Time Simulation Formatting selection to minutes and click OK. The transient reservoir data shows that the water in the discharge tank reached a height of 6 meters in 31 minutes. This is confirmed by the event message indicating the pump transient also initiated at this time (Figure 4).

The Quick Graph plot (Figure 3) shows the tank liquid height over time. To create this graph, go to the Reservoir Transient tab in the top section of the window, then right-click on any of the Liq. Height cells for the Discharge Reservoir, junction J9, and select the Quick Graph option. There is an increase in the tank liquid height until the time when the pump turns off. The graph then shows that there is actually back flow into the system, due to the hydrostatic pressure of the fluid in the tank. This explains why there is a Critical Warning, which states that the pump head cannot be predicted due to reverse flow through the pump. A check valve would need to be added to the system to prevent this situation in the case of pump shutdown.

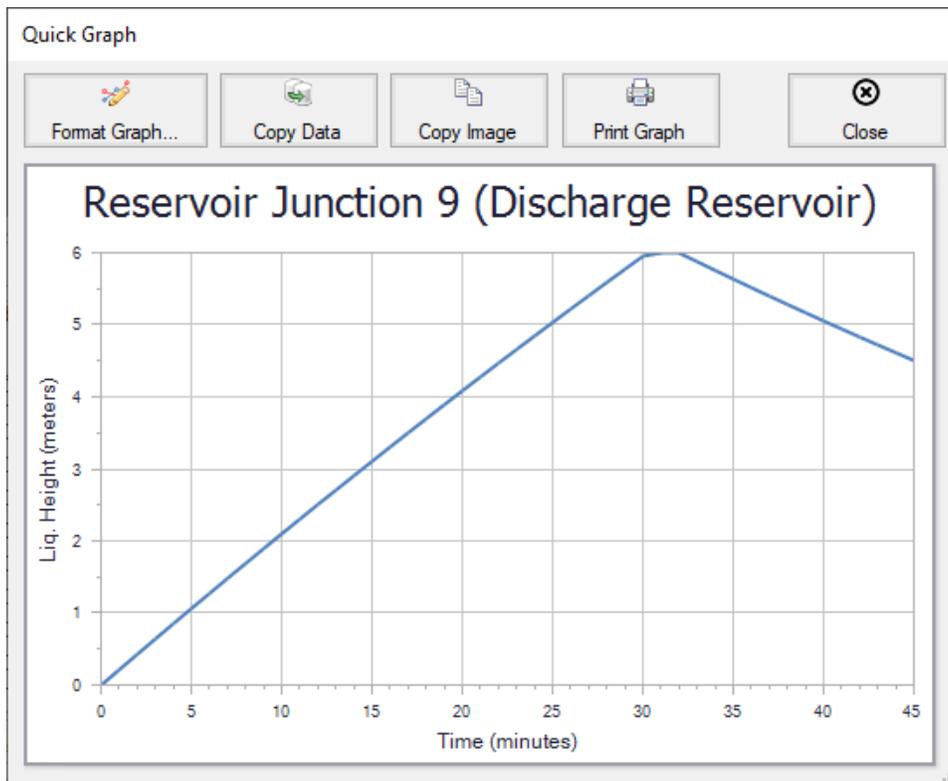


Figure 3: The liquid level in a finite tank junction can change with time

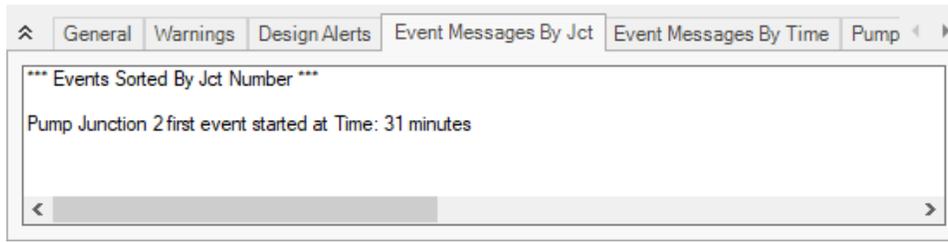


Figure 4: Event transients occurrences are recorded in the General section of the Output Window

Housing Project - XTS

Summary

This example uses the XTS module to simulate a supply pump used to replenish a water storage tank that is used to supply water to a housing development. There is a requirement that the height of the liquid in the water supply tank is not to drop below 3 meters. Use the flow conditions specified in the **4 inch Pipe** scenario (not the fire flow scenarios) of the Housing Project walk-through example as the basis for this example.

Note: This example can only be run if you have a license for the XTS module.

Topics Covered

- Using Dual Cyclic Junction Transients
- Understanding Transient Output
- Resolving Time Step issues

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the [Beginner - Filling a Tank - XTS](#) example, and is familiar with the basics of XTS transient analysis.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *Metric - Housing Project.fth*
- *Metric - Housing Project - XTS.fth*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

Housing Project - XTS

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **Metric - Housing Project.fth** example file. Right-click on the 4 inch Pipe scenario and select Save Scenario to File Without Children and save this scenario to a new model file.

The Workspace should look like Figure 1 below:

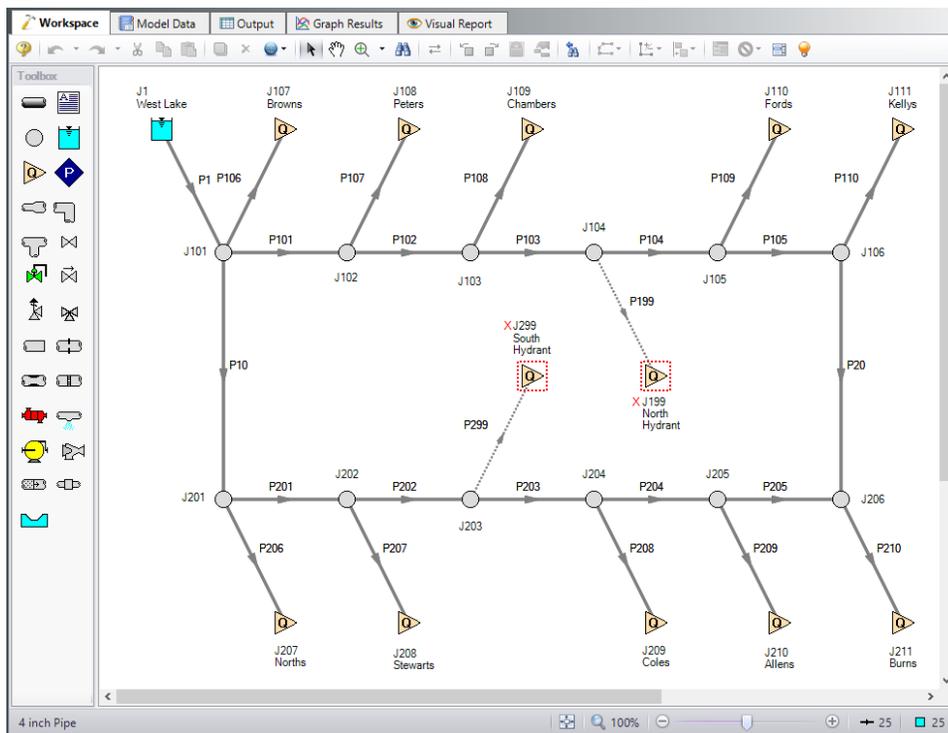


Figure 1: Workspace for XTS Water to Housing Project Example

For this example, we will be changing the West Lake junction (J1) from an infinite reservoir to a finite water supply tank. We will then add a new supply reservoir, as well as a supply pump and a check valve to the system. The new supply pump will replenish the supply tank as the water level in the tank is depleted.

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate XTS and select Transient to enable the XTS module for use. A new group will appear in Analysis Setup titled Transient Control. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Time Simulation. From here you can quickly toggle between Steady Only mode (normal AFT Fathom) and Transient (XTS mode).

Step 4. Modify the existing model

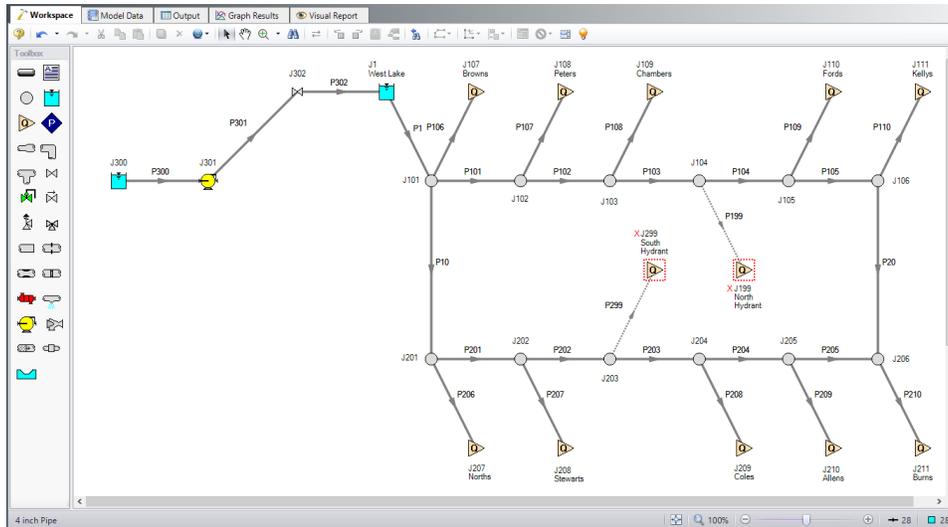


Figure 2: Workspace for XTS water to Housing Project example with supply pump system added

Modify the workspace to include the new supply reservoir, supply pump, check valve, and piping as shown in Figure 2. You may need to move the objects to the right on the Workspace to create room for the new pipes and junctions. Do this by selecting all pipes and junctions by either using CTRL+A or click-and-drag and selection box encompassing all objects. Then click, hold, and drag the objects to the right. Now define the new objects.

Pipe Properties

1. Pipe Model tab
 - a. Pipe Material = PVC - ASTM
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 4 inch
 - d. Type = schedule 40
 - e. Friction Model Data Set = Standard
 - f. Lengths =

Pipe	Length (meters)
300	7.5
301	150
302	1.5

Junction Properties

1. J300 Reservoir
 - a. Name = West Lake
 - b. Tank Model = Infinite Reservoir
 - c. Liquid Surface Elevation = 0 meters
 - d. Liquid Surface Pressure = 0 barG (0 kPa (g))
 - e. Pipe Depth = 0 meters

2. J301 Pump
 - a. Inlet Elevation = 0 meters
 - b. Pump Model = Centrifugal (Rotodynamic)
 - i. Analysis Type = Pump Curve
 - ii. Enter Curve Data =

Volumetric	Head
m3/hr	meters
0	300
45	350
90	240

- iii. Curve Fit Order = 2
 - c. Optional tab
 - i. Special Condition = Pump Off With Flow Through
1. J302 Check Valve
 - a. Inlet Elevation = 60 meters
 - b. Loss Model = Cv
 - c. Cv = 10
 - d. Forward Velocity to Close Valve = 0 meters/sec
 - e. Delta Pressure/Head to Reopen = Pressure
 - f. Delta Pressure = 0 bar (kPa)

2. J1 Reservoir
 - a. Name = Supply Tank
 - b. Tank Model = Finite Open Tank
 - c. Known Parameters Initially = Both Parameters (Liquid Surface Level & Surface Pressure)
 - d. Cross-Sectional Area = Constant, 6.5 meters²
 - e. Liquid Level Specification = Height from Bottom
 - f. Liquid Height = 4.9 meters
 - g. Liquid Surface Pressure = 0 barG (0 kPa (g))
 - h. Tank Height = 5 meters

- i. Pipe Depth = 5 meters
- j. Tank Bottom Elevation = 60 meters

Step 5. Define the Transient Control Group

Open Analysis Setup again and navigate to the Simulation Duration panel, part of the Transient Control group.

For this example, with a Start Time of 0 minutes, set the Stop Time to 120 minutes with a time step of 1 minute. The output will be saved to the output file for every time step.

Step 6. Specify the Time Simulation Formatting

Open the Output Control window and select the Format & Action tab. Set the Time Simulation Formatting units to **minutes** and click OK to save and close the window.

Step 7. Enter the Pump Transient Data

For this example, the pump will be set to turn on when the liquid level in the supply tank drops below 3 meters, and turn off when the water level returns to 5 meters. To do this, we will use a Dual Event Cyclic event transient.

The Dual Event Cyclic event requires that two transients be defined. In this case, the first transient will be the pump turning on when the water level drops below 3 meters, and the second transient will be the pump turning off when the water level returns to 5 meters. The pump speed transients need to be included for both transient events as well. Enter the following in the pump properties window.

- 1. Transient tab
 - a. Transient Type = Speed vs. Time
 - b. Transient Special Condition = None
 - c. Initiation of Transient = Dual Event Cyclic
 - d. First Event
 - i. Event Type = Reservoir Liquid Height
 - ii. Condition = Less Than or Equal To
 - iii. Value = 3 meters
 - iv. Junction = 1 (Supply Tank)
 - e. First Transient =

Time (seconds)	Speed (Percent)
0	0
30	100

- e. Second Event
 - i. Event Type = Reservoir Liquid Height
 - ii. Condition = Greater Than or Equal To
 - iii. Value = 5 meters
 - iv. Junction = 1 (Supply Tank)

e. Second Transient =

Time (seconds)	Speed (Percent)
0	100
30	0

The screenshot shows the 'Pump Properties' dialog box with the 'Transient' tab selected. The 'Event Type' is set to 'Reservoir Liquid Height', the 'Condition' is 'Less Than or Equal To', the 'Value' is '3 meters', and the 'Junction' is '1 (Supply Tank)'. The 'Transient Data' table is visible, showing two data points: (0, 100) and (30, 0). The 'Speed vs. Time' graph shows a step function that drops from 100% at 0 seconds to 0% at 30 seconds.

Figure 3: First Event and First Transient Data for Water to Housing Project XTS example

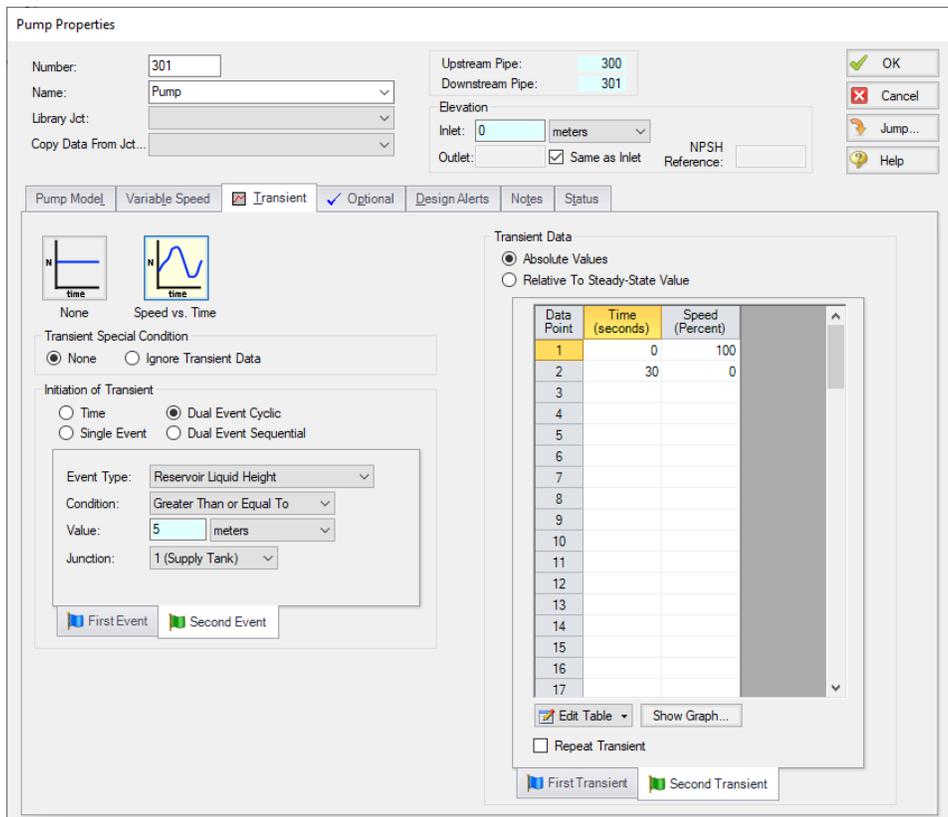


Figure 4: Second Event and Second Transient Data for Water to Housing Project XTS example

For this example, at the beginning of the simulation, the storage tank is full, so the pump will not be running initially. To account for this, the pump Special Condition on the Optional Tab must be set to Pump Off With Flow Through. In order to be consistent, the First Transient defined must be defined with an initial speed of 0, indicating a pump startup.

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 8. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 9. Examine the Output

Figure 5 shows a Quick Graph plot of the liquid height in the J1 Supply Tank. The plot illustrates how the Dual Event Cyclic transient will repeat continuously as the event to trigger each transient is met, in turn. The plot shows the tank draining due to the flow to the housing project. When the liquid level reaches 3 meters, the pump starts, causing the liquid level in the tank to rise again. When the level reaches 5 meters, the transient to stop the pump is initiated. Because the liquid level reaches the top of the tank at this time, the Warnings tab will display a warning about the tank overflowing at 73 minutes. With the pump turned off, the tank drains until the level drops to 3 meters and the pump is started again.

The plot shows the liquid level dropping below 3 meters. This is due, primarily with the time step resolution that was set in Transient Control. The time step was set to 1 minute, but the pump startup transient is 30 seconds. The size of the time step leads to *overshoot* in the calculations. Open the Simulation Duration panel, and set the time step to 10 seconds. Set the output to save to file every 6th time step (write output once every minute). The results from this case are shown in Figure 6. The results using the 10-second time step clearly show the effect of using the smaller time step.

The user should always carefully consider the time step being used in relation to the length of the transient events defined for their analysis.

Note: You may notice a warning message indicating that the static pressure is lower than vapor pressure at the outlet of pipe 301. Since this example is for demonstration purposes only, this warning may be ignored. However, warnings should otherwise never be ignored.

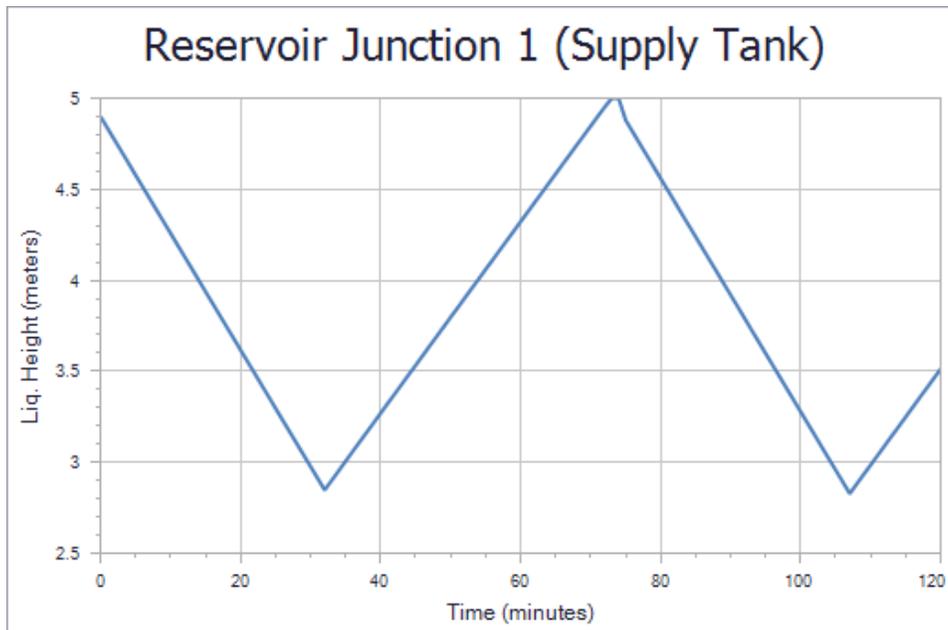


Figure 5: Plot of Supply Tank liquid level showing Dual Event Cyclic transient

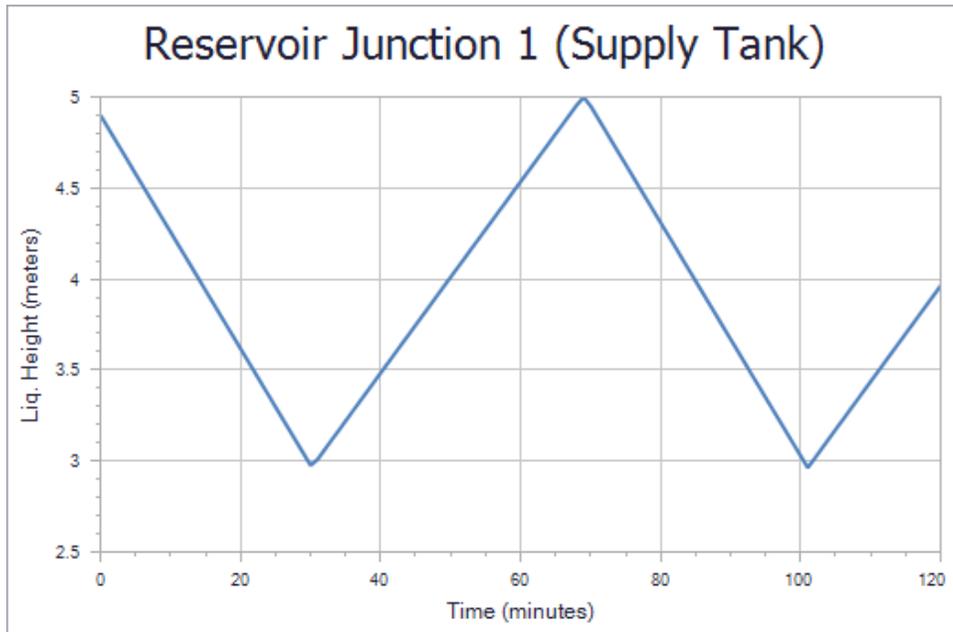


Figure 6: Plot of Supply Tank liquid level with 10-second time step

Variable Demand - XTS

Summary

This example uses a pump transient to initiate auxiliary pump start up for a system with variable demands to ensure adequate pressure drop across a control valve. Additional loads will be added to the system every 30 seconds, until the control valve open percentage exceeds 50%. At that time, the auxiliary pump will start to handle the additional capacity.

Note: This example can only be run if you have a license for the XTS module.

Topics Covered

- Defining Control Valve Event Transient
- Understanding Transient Output
- Using Transient Graphs

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the [Beginner - Filling a Tank - XTS](#) example, and is familiar with the basics of XTS transient analysis.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- *Metric - Variable Demand - XTS.fth*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate XTS and select Transient to enable the XTS module for use. A new group will appear in Analysis Setup titled Transient Control. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Time Simulation. From here you can quickly toggle between Steady Only mode (normal AFT Fathom) and Transient (XTS mode).

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = AFT Standard
 - b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
 - c. Temperature = 21 deg. C

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

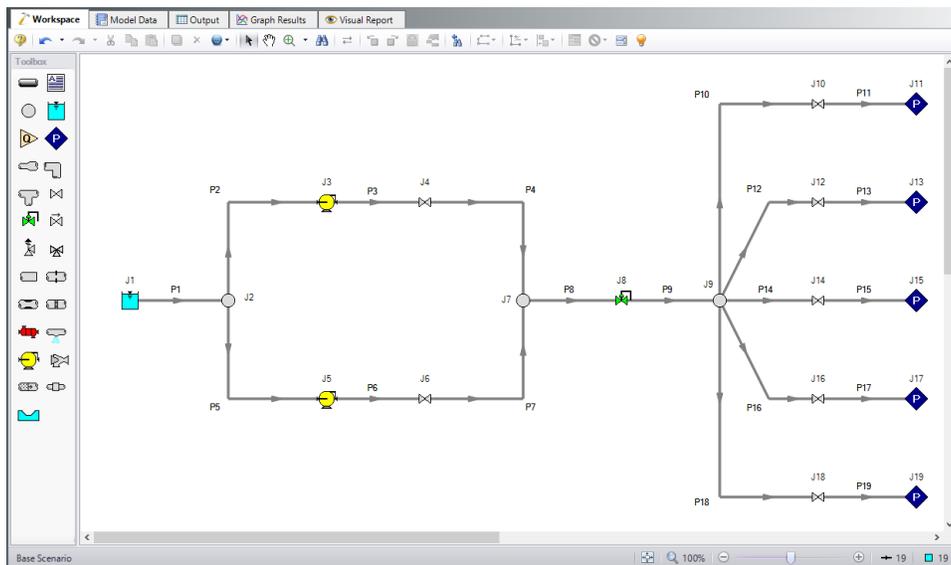


Figure 1: Workspace for XTS Variable Demand example

The system is in place but now we need to enter the properties of the objects. Double-click each pipe and junction and enter the following properties. The required information is highlighted in blue.

Pipe Properties

1. Pipe Model tab
 - a. Pipe Material = PVC - ASTM
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 4 inch
 - d. Type = schedule 40
 - e. Friction Model Data Set = Standard
 - f. Lengths =

Pipe	Length (meters)
1-7 & 10-19	3
8 & 9	15

Junction Properties

1. J1 Reservoir
 - a. Tank Model = Finite Open Tank
 - b. Known Parameters Initially = Both Parameters
 - c. Cross-Sectional Area = Constant, 2 meters²
 - d. Liquid Level Specification = Height from Bottom
 - e. Liquid Height = 3 meters
 - f. Liquid Surface Pressure = 0 barG (0 kPa (g))
 - g. Tank Height = 3 meters
 - h. Pipe Depth = 3 meters
 - i. Tank Bottom Elevation = 0 meters
2. J2, J7, & J9 Branches
 - a. Elevation = 0 meters
3. J3 & J5 Pumps
 - a. Inlet Elevation = 0 meters
 - b. Pump Model = Centrifugal (Rotodynamic)
 - c. Analysis Type = Pump Curve
 - d. Pump Curve Data =

Volumetric	Head
m ³ /hr	meters
0	24

Volumetric	Head
m3/hr	meters
28	27
57	15

- e. Curve Fit Order = 2
- 4. J4 & J6 Valves
 - a. Inlet Elevation = 0 meters
 - b. Valve Data Source = User Specified
 - c. Loss Model = Cv
 - d. Loss Source = Fixed Cv
 - e. Cv = 100
- 5. J8 Control Valve
 - a. Inlet Elevation = 0 meters
 - b. Control Valve Model tab
 - i. Valve Type = Pressure Reducing (PRV)
 - ii. Always Control (Never Fail) = Unchecked
 - iii. Control Setpoint = Pressure, Static
 - iv. Setpoint = 3.1 bar (310 kPa)
 - v. Loss When Fully Open = Cv
 - vi. Loss Source = From Open % Table (on Optional Tab)
 - c. Optional tab
 - i. Open Percentage Data =

Open Pct.	Cv
0	0
100	100

- 6. J10, J12, J14, J16, & J18 Valves
 - a. Inlet Elevation = 3 meters
 - b. Valve Data Source = User Specified
 - c. Loss Model = Cv
 - d. Loss Source = Fixed Cv
 - e. Cv = 10
- 7. J11, J13, J15, J17, & J19 Assigned Pressures
 - a. Elevation = 3 meters
 - b. Pressure = 0 barG (0 kPa (g))
 - c. Pressure Specification = Stagnation

Step 5. Define the Transient Control Group

Open Analysis Setup again and navigate to the Simulation Duration panel, part of the Transient Control group.

For this example, with a Start Time of 0 minutes, set the Stop Time to 3 minutes with a time step of 5 seconds. The output will be saved to the output file for every time step.

Step 6. Enter the Transient Data

Valve Transients

Initially, the only flow through the system will be through the J10 Valve. Valves J12, J14, J16, and J18 will start closed, and they will open 30 seconds apart, in sequence. Set the Special Condition for Valves J12, J14, J16, and J18 to Closed on the Valve Properties Optional tab.

The valve opening transient for Valve J12 is shown in Figure 2. Valves J14, J16, and J18 should be set up with the same opening transient, and by adding 30 seconds to the Time Absolute event for each valve so that valve J14 opens at 60 seconds, valve J16 opens at 90 seconds, and valve J18 opens at 120 seconds.

Note that a red XT will now be displayed next to the junction labels to show that these valves are closed, and have transients applied to them.

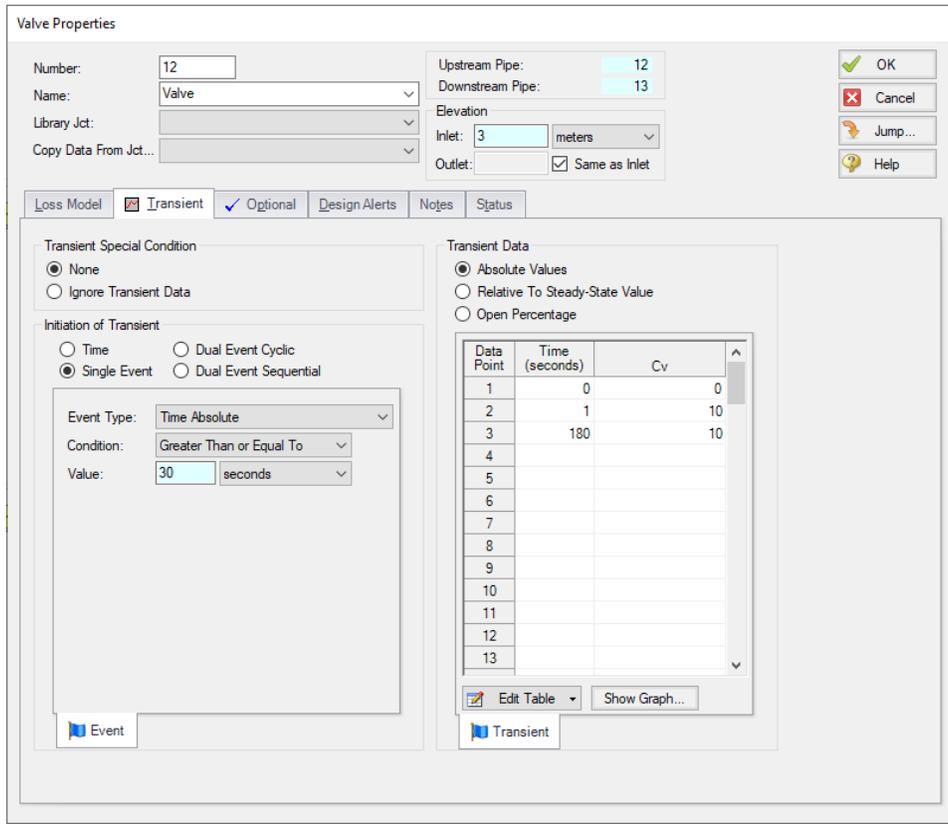


Figure 2: Valve J12-J18 opening transient profiles are defined on the Transient tab on the Valve Properties window

Pump Transient

Pump J5 is not operating at the beginning of this simulation. On the Optional tab of the Pump Properties window set the Pump J5 Special Condition to Pump Off With Flow Through. The pump is set to start when the open percentage of Control Valve J8 exceeds 50%. The transient event definition and pump startup profile are shown in Figure 3.

Variable Demand - XTS

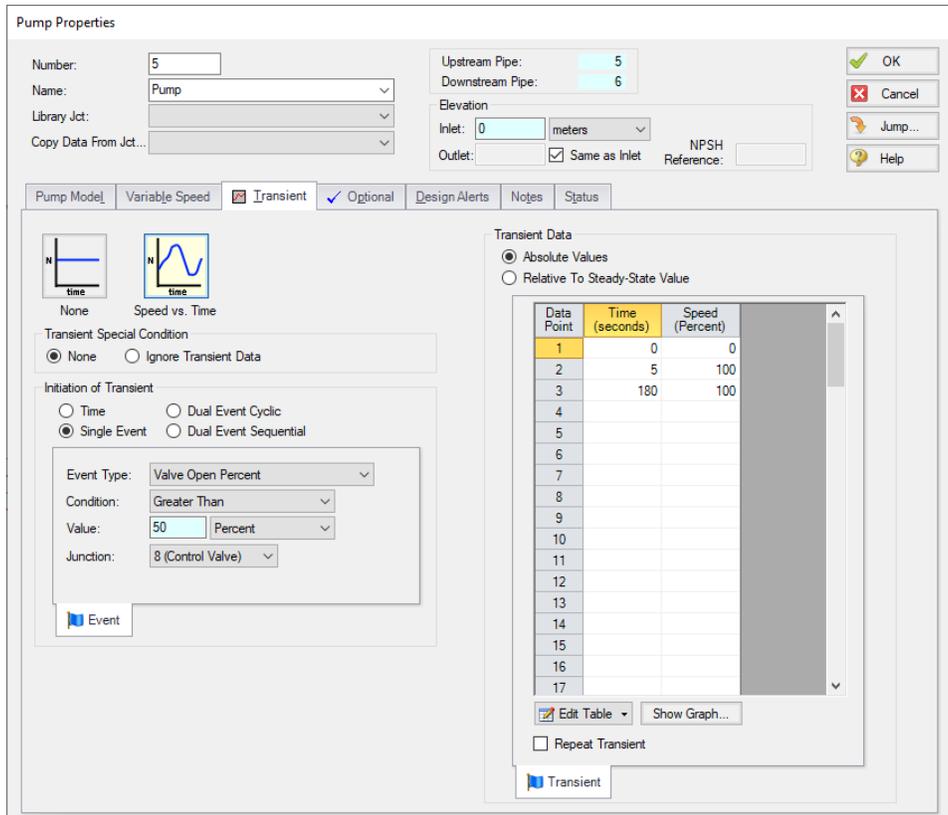


Figure 3: Pump J5 transient startup profile is defined on the Transient tab on the Pump Properties window

Isolation Valve J6 should be closed initially to prevent flow through the leg while Pump J5 is not operating. Do this by setting the Valve J6 Special Condition to Closed on the Optional tab. Valve J6 will open when Pump J5 starts. To do this, use the same transient event as defined for Pump J5. This is shown in Figure 4.

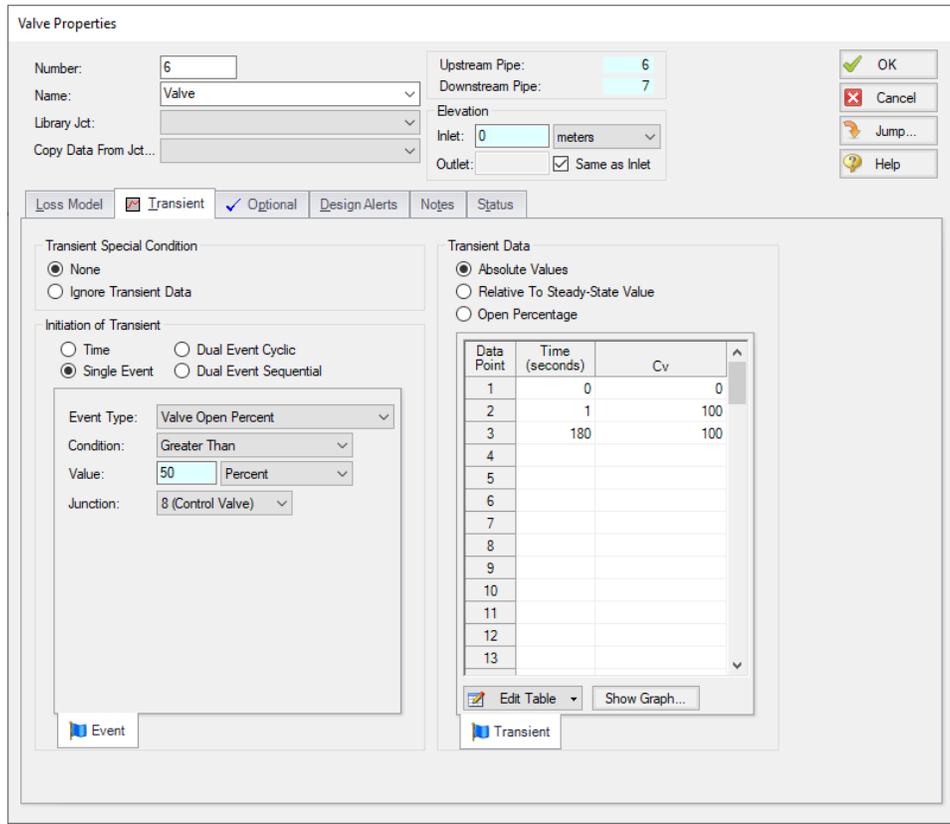


Figure 4: Valve J6 opening transient profile is defined on the Transient tab on the Valve Properties window

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 7. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 8. Examine the Output

The occurrence of the transient events is recorded on two tabs in the General Output section. The first tab sorts the events by junction, and the second tab sorts the events chronologically. Figure 5 shows the chronological ordering of the transient events for this run. This output shows the downstream valves opening at 30-second intervals. It also shows that Pump J5 started 65 seconds into the run, and the isolation valve J6 also opened at 65 seconds.

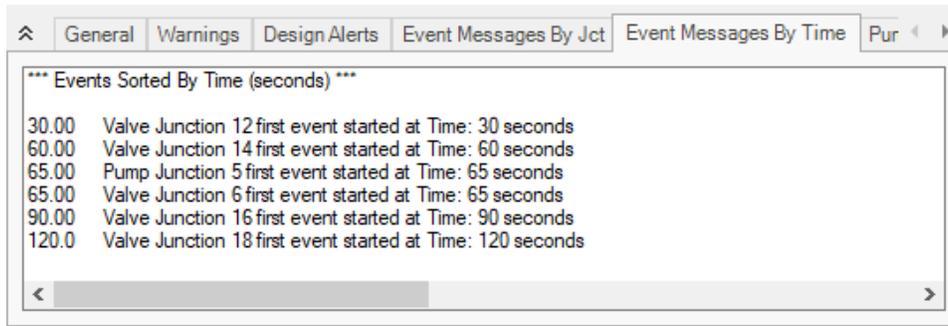


Figure 5: A chronological list of Transient Events is shown in the General Output section of the Output Window

Plotting the speed of Pump J5 vs. time as a Quick Graph from the Output shows the pump starting up at 65 seconds, and coming up to full speed over 5 seconds. This is shown in Figure 6.

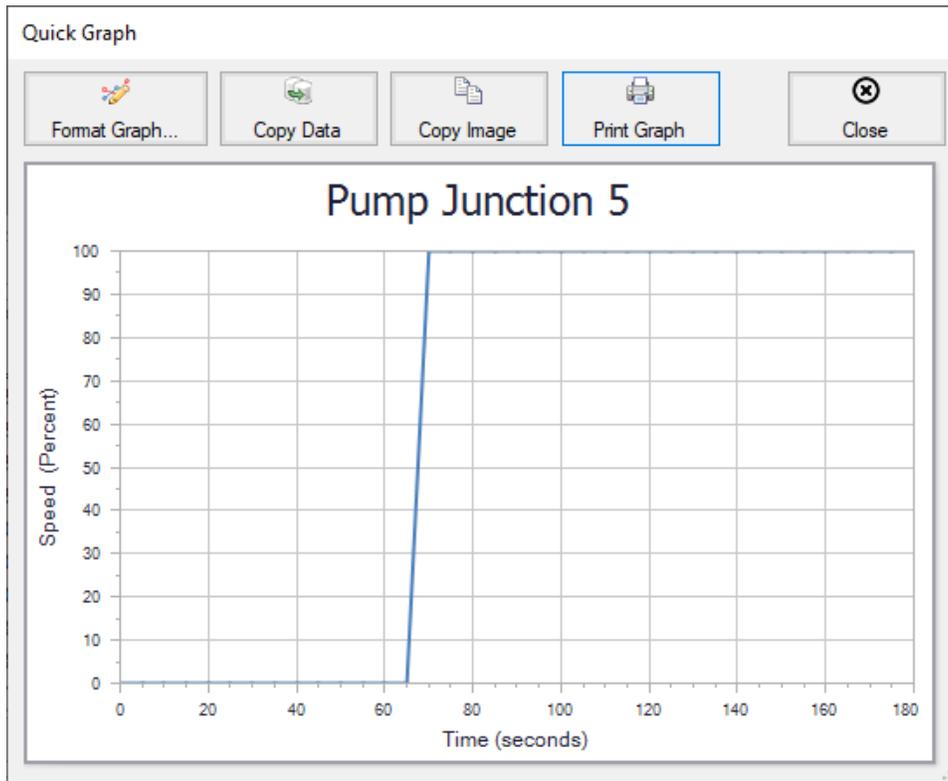


Figure 6: Quick Graph can be used to plot the Pump J5 Speed vs. Time

Now go to the Graph Results window. A stacked graph can be made for the Control Valve J8 as shown in Figure 7 to see the affect of the opening valves on the flow rate at the control valve, as well as how the open percent at the control valve responds. It can be seen here that the control valve open percent reaches 50% at 65 seconds, which correlates with the beginning of the pump startup.

To create these stacked graphs, go to the Transient tab in Graph Control. Change the Pipes or Junctions selection to Valves and check the box next the 8, Control Valve. At the bottom, click the Add button next to Select Parameter to add a second parameter to graph. Set the first parameter to Volumetric Flow Rate with m³/hr Units, and set the second parameter to Open Percentage with Percent Units. Now click Generate.

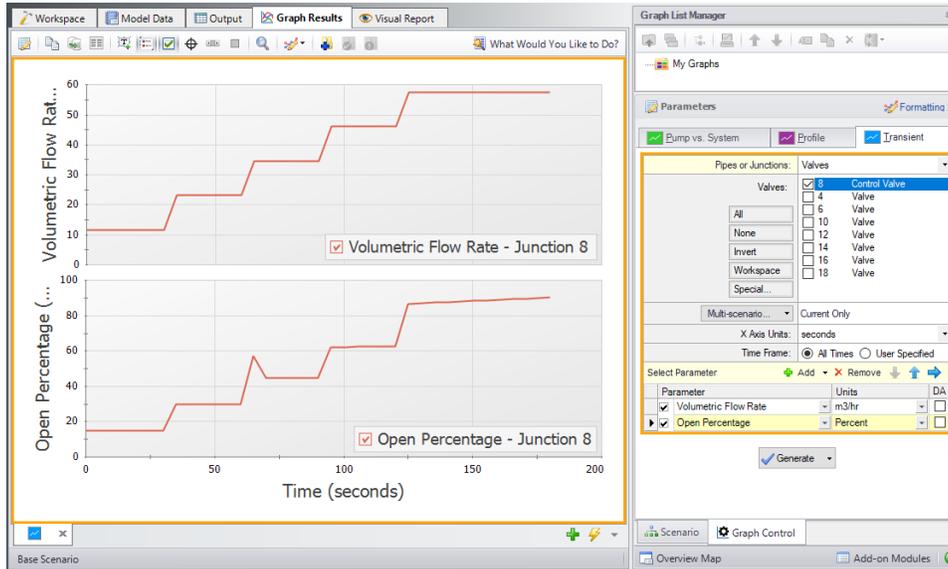


Figure 7: Stacked graph on the Graph results tab showing the Volumetric Flow Rate and Open Percentage over time at Control Valve J8

Settling Slurry (SSL) Examples

The examples below utilize the Settling Slurry module.

Fathom SSL Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
Beginner - Pump Sizing for Sand Transfer System	Beginner	Slurry	3	1	Learn the basics of the SSL module such as defining a Detailed slurry calculation model in Analysis Setup, using the Slurry De-Rating feature in the Pump Properties window, adding Slurry parameters using the Output Control window, and generating Slurry System Curves in the Graph Results window.
Slurries with Variable Fluid Properties	Intermediate	Water/Slurry	2-3	1	Learn how to use the SSL module with Variable Fluid Properties. See how to use the Volume Balance junction and the Simplified slurry calculation model.
Slurry System Feasibility Study	Advanced	Slurry	1	0	Learn how to use the SSL module and the Scenario Manager to determine the feasibility of selected operating conditions. See how to use the Multi-Scenario output feature in the Output window and how to perform a Batch run for a model with multiple scenarios.

Beginner - Pump Sizing for Sand Transfer System - SSL

Summary

This example shows how SSL can be used to size a slurry pump as part of a system design process. The piping for a slurry system that moves sand is being designed. The system will pump 25% sand by volume from an open supply vessel with a liquid surface elevation of 1.5 meters to an open receiving tank with a liquid surface elevation of 3 meters.

Use the SSL module to size the pump.

Note: This example can only be run if you have a license for the SSL module.

Topics Covered

- Entering solids data
- Reviewing calculated slurry output
- Creating slurry system curves

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Pump Sizing for Sand Transfer System - SSL.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate SSL and select Settling to enable the SSL module for use. The items in the Fluid Properties group will change to accommodate slurry definitions. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Slurry. From here you can quickly toggle between Disable mode (normal AFT Fathom) and Settling (SSL mode).

Step 3. Define the Fluid Properties Group

Open Analysis Setup from the toolbar or from the Analysis menu and input the following on the respective panels.

1. Carrier Fluid (Figure 1)
 - a. Fluid Model = Basic (Water)
 - b. Temperature = 21 deg. C
2. Slurry Definition (Figure 2)
 - a. Slurry Model = Settling Slurry - Wilson, Addie, Clift
 - b. Slurry Calculation Method = Detailed
 - c. Concentration Type = Volume Fraction
 - d. Amount Solids Added = 25 Percent
3. Solids Definition (Figure 3)
 - a. Solids Specifications = User Specified Solids Added
 - b. M (Stratification Ratio Exponent) = Calculated
 - c. Terminal Velocity Parameter, $V_t / V_{ts} (X_i) = 1$
 - d. Density = 2.9 S.G. water
 - e. $d_{50} = 0.0381$ cm
 - f. $d_{85} = 0.0762$ cm

Settling Slurry (SSL) Examples

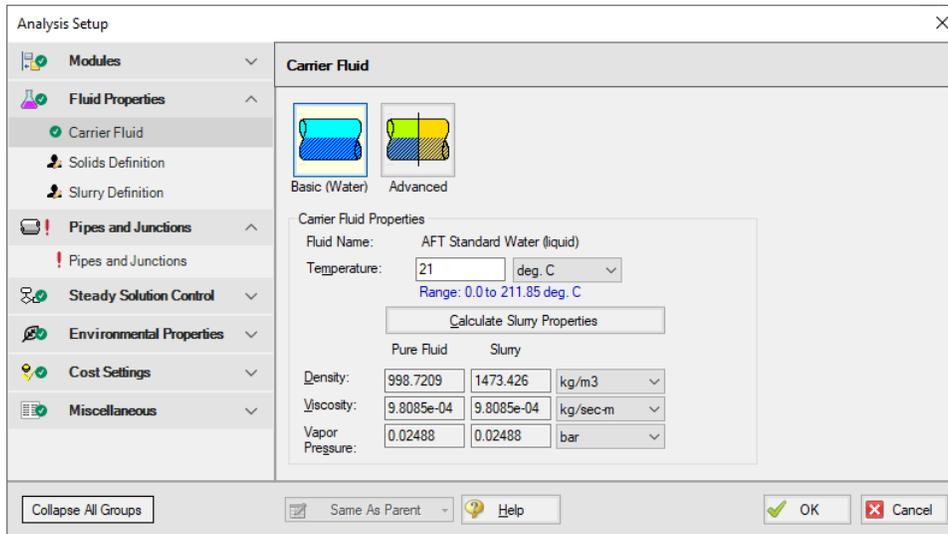


Figure 1: Carrier Fluid panel in the Fluid Properties group in Analysis Setup with SSL activated

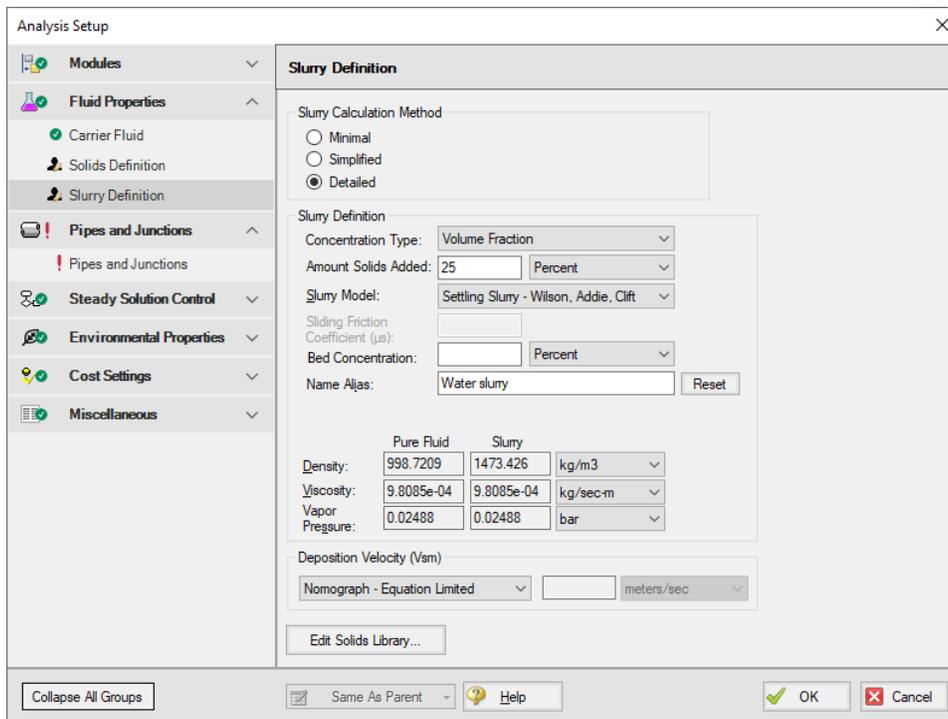


Figure 2: Slurry Definition panel in the Fluid Properties group in Analysis Setup with SSL activated

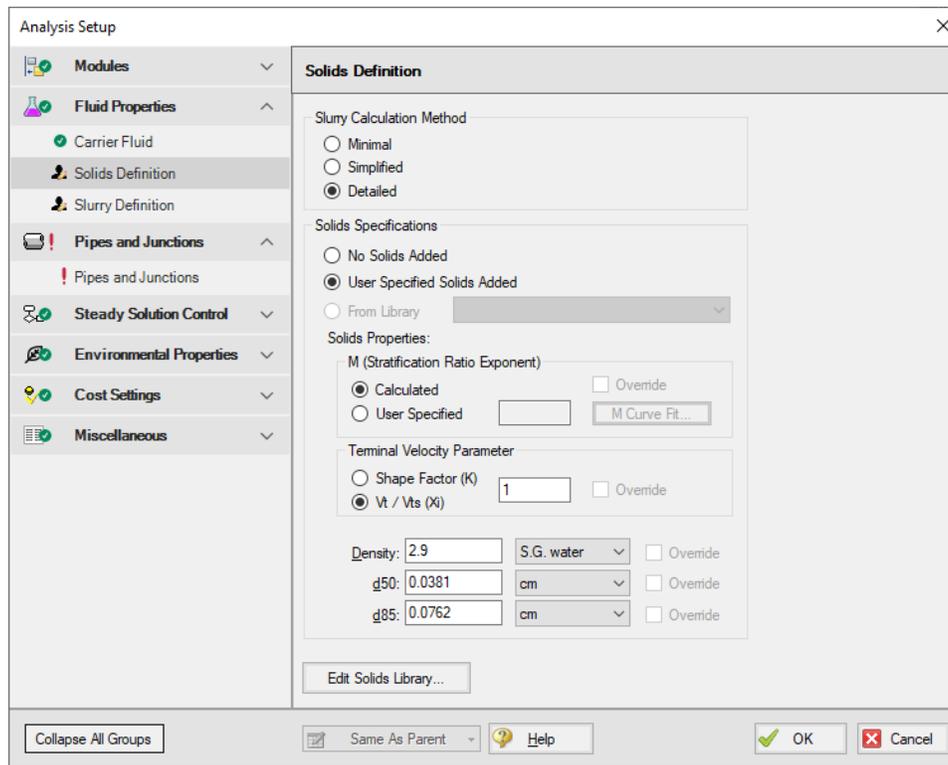


Figure 3: Solids Definition panel in the Fluid Properties group in Analysis Setup with SSL activated

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

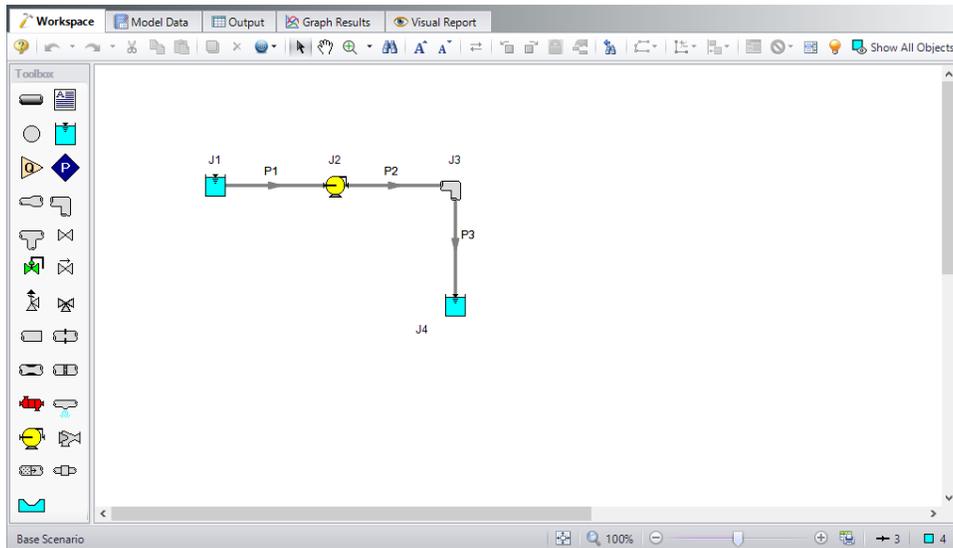


Figure 4: Layout of pipe system for Pump Sizing for Sand Transfer System example

Enter the following pipe and junction properties.

Pipe Properties

1. All Pipes
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 12 inch
 - d. Type = STD
 - e. Friction Model Data Set = Standard
 - f. Length =

Pipe	Length (meters)
1	6
2	150
3	15

Junction Properties

1. J1 Reservoir
 - a. Name = Supply Tank
 - b. Liquid Surface Elevation = 1.5 meters

- c. Liquid Surface Pressure = 0.7 barG (70 kPa(g))
- d. Pipe Depth = 1.5 meters
- 2. J2 Pump
 - a. Inlet Elevation = 0 meters
 - b. Pump Model tab
 - i. Pump Model = Centrifugal (Rotodynamic)
 - ii. Analysis Type = Sizing
 - iii. Parameter = Mass Flow Rate
 - iv. Fixed Flow Rate = 1270 m-ton/hr Solids
 - c. Slurry De-Rating tab
 - i. De-Rating Method = ANSI/HI Standard 12.1-12.6-2016
 - ii. Percent Fines = 0 %
 - iii. Head De-Rating Data = Specify Impeller Size Directly
 - iv. Impeller Size = 20 cm
- 3. J3 Elbow
 - a. Inlet Elevation = 0 meters
 - b. Type = Standard Elbow (knee, threaded)
 - c. Angle = 90 Degrees
- 4. J4 Reservoir
 - a. Name = Receiving Tank
 - b. Liquid Surface Elevation = 3 meters
 - c. Liquid Surface Pressure = 2 barG (200 kPa(g))
 - d. Pipe Depth = 3 meters

To show the names of the reservoirs, a new Workspace Layer will be created.

1. Create a new layer by selecting New Layer and then Blank Layer from the Workspace Layers tab of the Quick Access Panel.
2. Name the layer Reservoir Labels.
3. Edit the layer by selecting it and clicking the gear icon to bring up the Layer Settings window.
4. Open the Show/Hide Labels panel and uncheck Force Shown Labels to Match Shown Objects.
5. Toggle the visibility icon next to the reservoirs to on. This will cause their labels to be shown.
6. Navigate to the Junction Parameters panel and expand the Commonly Used Junction Parameters list.
7. Double-click Junction Name to add it to the list on the right-hand side. This will add the Junction Name to the label of the junctions that were defined in the Show/Hide Labels panel.
8. Close the Layer Settings window.

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

Open the **Output Control** window by selecting Output Control from the Output toolbar or Tools menu.

On the Display Parameters tab, click the **Pumps** button, and add the De-rating Head Correction factor (**CH**) to the Pump Summary output.

The Output Control window can also be used to configure the slurry output data, as is shown in Figure 5. Click OK when you are finished.

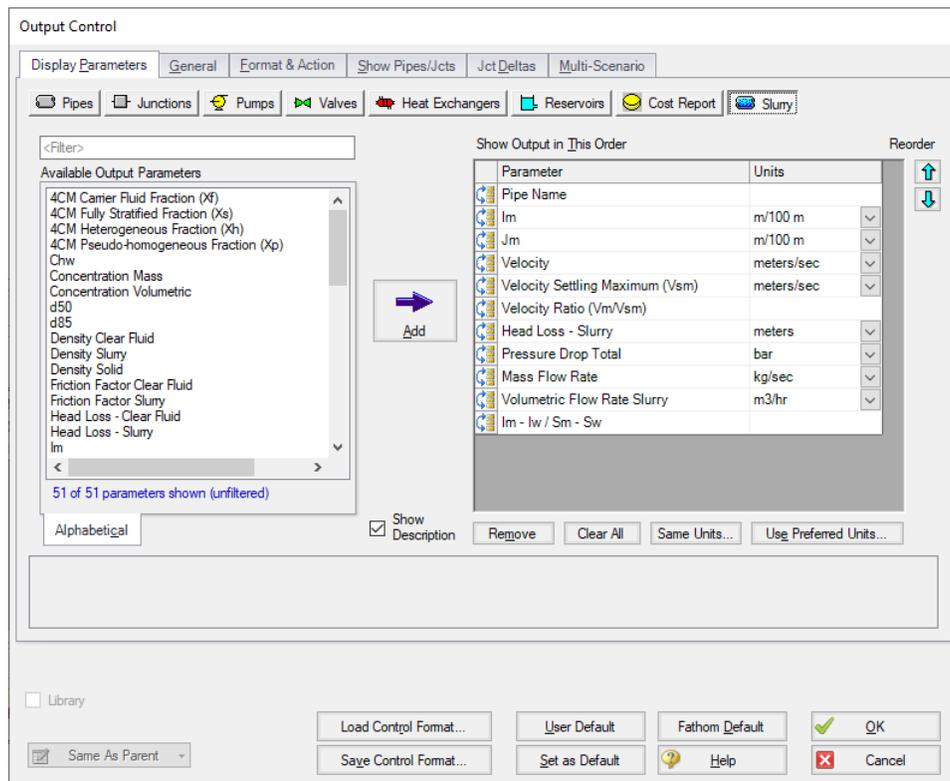


Figure 5: The Output Control window can be used to configure the slurry output data

The Slurry tab in the pipes section shows the slurry results, as shown in Figure 6. Here one can see that the velocity is 6.674 meters/sec, which is safely above the 3.498 meters/sec max settling velocity. Other pressure drop and relevant slurry data can be viewed here.

Settling Slurry (SSL) Examples

Also shown in Figure 6 is the Pump Summary in the General Section. There, the required pump head is shown as 27.81 meters, and the de-rating head correction factor is 0.7887. Hence a pump should be selected that can generate $27.81/0.7887 = 35.26$ meters of head based on water.

General														
Jct	Results Diagram	Name	Vol. Flow (m3/hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)	CH
2	Show	Pump	1,753	717.6	4.018	27.81	100.0	N/A	195.6	N/A	N/A	12.61	N/A	0.7887

Pipes											
Pipe	Name	Im (m/100 m)	Jm (m/100 m)	Velocity (meters/sec)	Settling Velocity Maximum (meters/sec)	Vm/Vsm	dH Slurry (meters)	dP (bar)	Mass Flow Rate (kg/sec)	Vol Flow Slurry (m3/hr)	(Im - Iw) / (Sm - Sw)
1	Pipe	14.17	9.607	6.674	3.498	1.908	0.5764	0.08331	717.6	1,753	0.08562
2	Pipe	14.17	9.607	6.674	3.498	1.908	14.4107	2.08207	717.6	1,753	0.08562
3	Pipe	14.17	9.607	6.674	3.498	1.908	1.4411	0.20815	717.6	1,753	0.08562

All Junctions									
Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)	
1	Supply Tank	1.713	1.930	1.713	1.930	1,753	717.6	0.0000	
2	Pump	1.518	5.537	1.847	5.865	1,753	717.6	0.0000	
3	Bend	3.455	3.327	3.783	3.655	1,753	717.6	0.3900	
4	Receiving Tank	3.013	3.447	3.013	3.447	1,753	717.6	0.0000	

Figure 6: The Output window shows the Slurry results table in the center Pipes section and the pump de-rating in the Pump Summary in the upper General section

Also of interest is the system curve. Select the Graph Results window tab then in the Graph Control tab in the Quick Access Panel select the Slurry System Curve tab. See Figure 7.

By default the parameter to graph is Jm vs. Velocity. Select the units of Jm as m/m that will make it completely dimensionless as is typically done. Check the box for Water System Curve which cross-plots the water curve and then click Generate. The curves are shown in Figure 8.

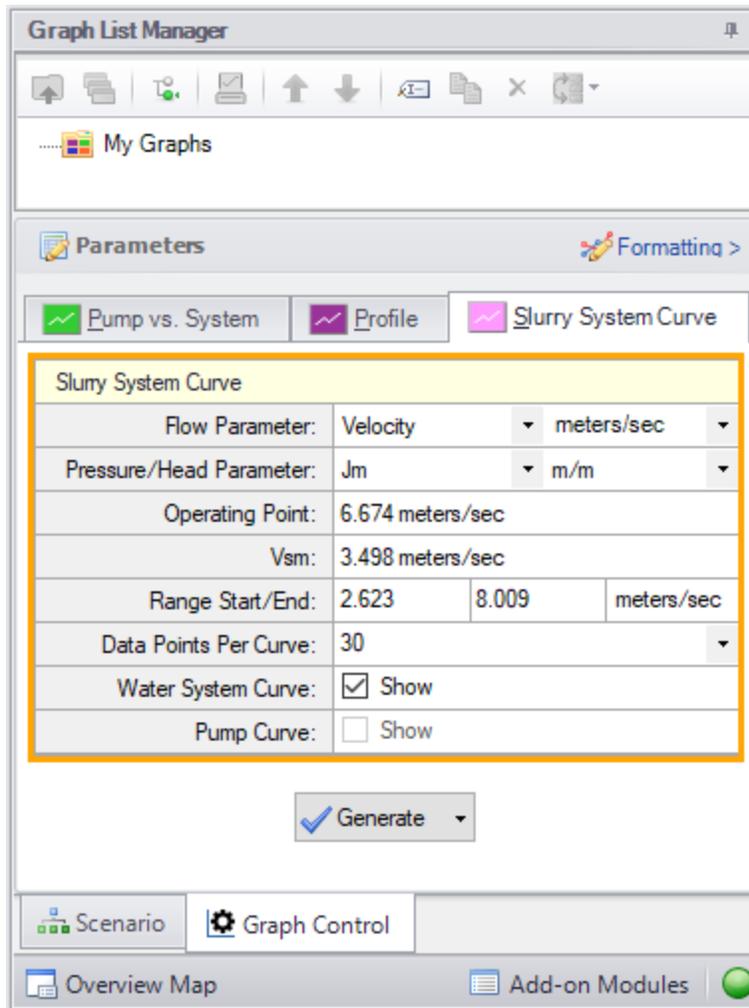


Figure 7: Data selection for graphing slurry system curve

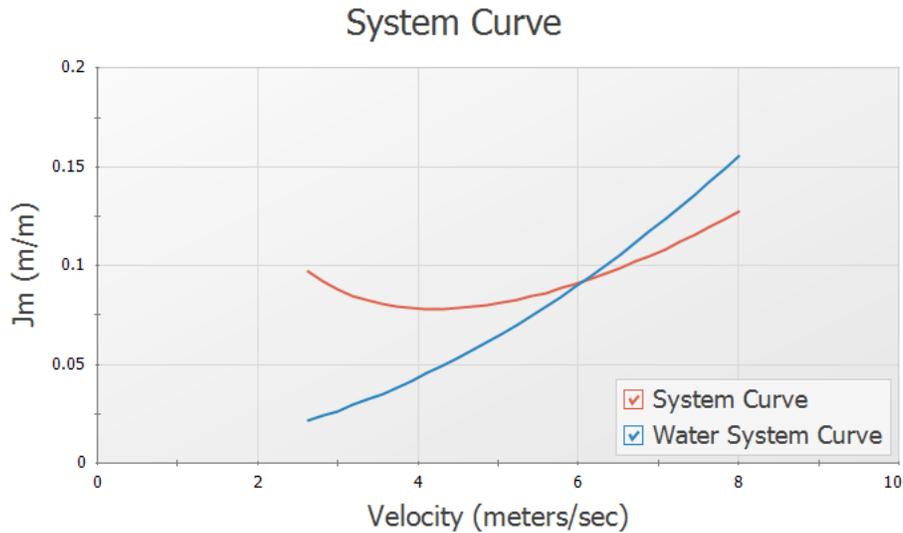


Figure 8: Slurry system curve and water curve in Graph Results window

Conclusion

The pump head requirements were determined, and a de-rating factor was calculated according to the ANSI/HI Standard 12.1-12.6-2016. The de-rating factor for pump head (CH) was used to manually calculate pump head using water, such that the correct size pump could be selected when looking at manufacturer data based on water curves. The velocity in each pipe was found to be higher than the max settling velocity (V_{sm}) of the sand and water slurry which ensured that no settling would occur. A slurry system curve was created to allow the user to see where the system was operating with respect to solids settling.

Slurries with Variable Fluid Properties - SSL

Summary

This example demonstrates the fundamental concepts of the Settling Slurry (SSL) add-on module by way of example. The example shows how SSL can be used to model the introduction of solids into a clear fluid stream.

Evaluate pump and system performance for a slurry system that will pump 30% sand by volume for three cases:

1. Pumping clear water (no solids in system)
2. Solids introduced into the system, and propagated through to the pump discharge
3. Solids have propagated through the entire system

This example is essentially modeling four snapshots in time as the slurry is injected and pumped through the pipeline. These four snapshots in time can be modeled through the usage of the [Volume Balance junction](#).

Note: This example can only be run if you have a license for the SSL module.

Topics Covered

- Entering solids data
- Using Volume Balance junctions with variable fluid properties
- Creating slurry system curves and evaluating pump performance

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through [Beginner - Pump Sizing for Sand Transfer System - SSL](#) example.

Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Fathom installation:

- **Metric - Slurries with Variable Fluid Properties - SSL.fth**

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate SSL and select Settling to enable the SSL module for use. The items in the Fluid Properties group will change to accommodate slurry definitions. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Slurry. From here you can quickly toggle between Disable mode (normal AFT Fathom) and Settling (SSL mode).

Step 3. Define the Fluid Properties Group

Open Analysis Setup from the toolbar or from the Analysis menu and input the following on the respective panels.

1. Carrier Fluid - Figure 1
 - a. Fluid Model = Advanced
 - b. Fluid Library = AFT Standard
 - c. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
 - d. Temperature = 4 deg. C
 - e. Select Variable Fluid Properties
2. Solids Definition - Figure 2
 - a. Slurry Calculation Method = Simplified
 - b. Solids Specifications = User Specified Solids Added
 - c. M (Stratification Ratio Exponent) = User Specified
 - d. Value = 1.7
 - e. Density = 2.65 S.G. water
 - f. d50 = 0.041 cm
3. Slurry Definition - Figure 3
 - a. Slurry Calculation Method = Simplified (this selection will be the same as on the Solids Definition panel)
 - b. Concentration Type = Volume Fraction
 - c. Amount Solids Added = 30 Percent

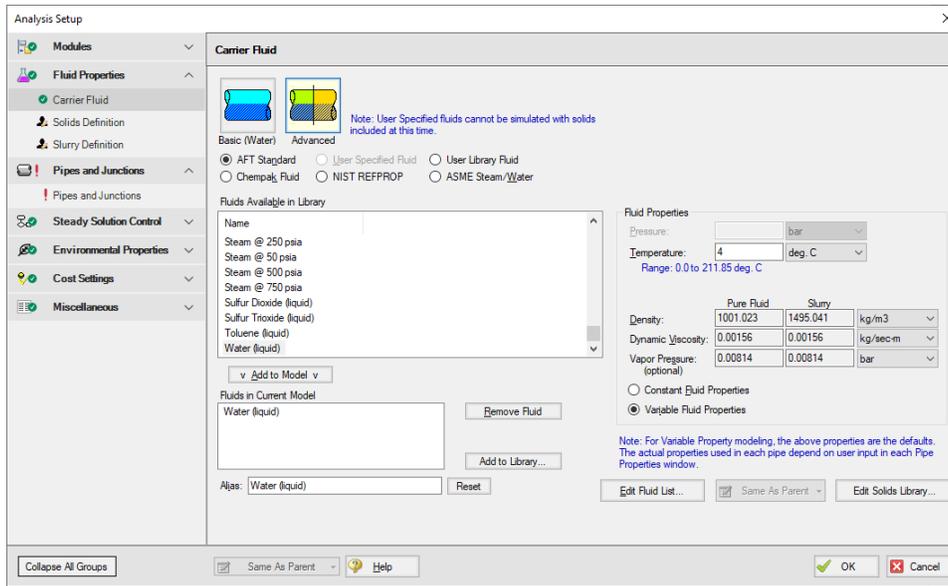


Figure 1: Carrier Fluid panel in the Fluid Properties group in Analysis Setup with SSL activated

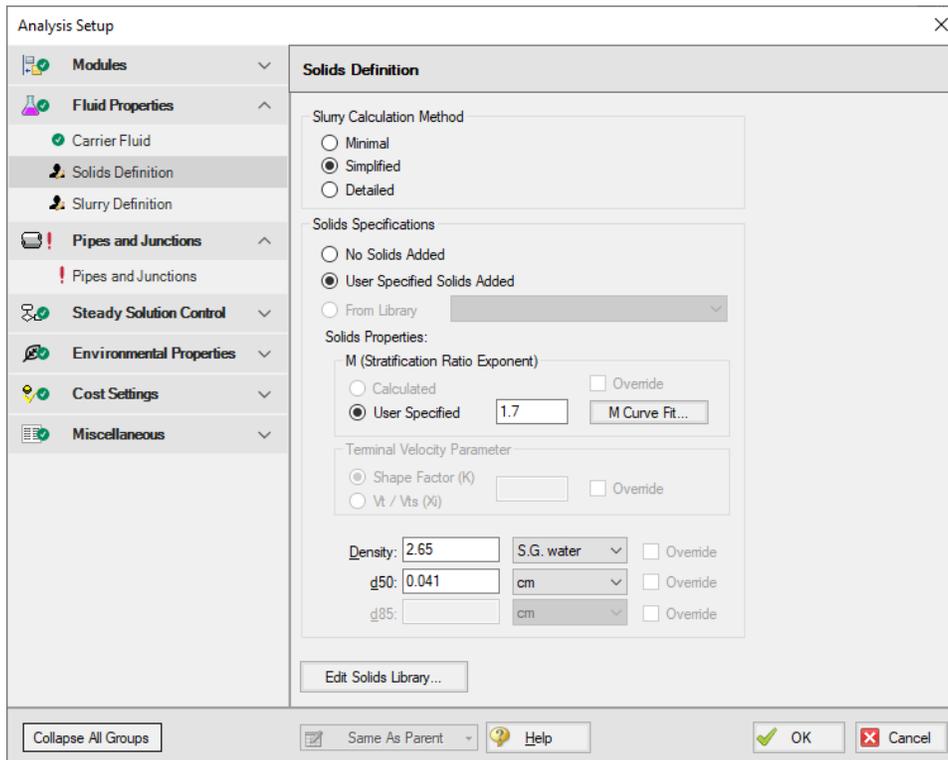


Figure 2: Solids Definition panel in the Fluid Properties group in Analysis Setup with SSL activated

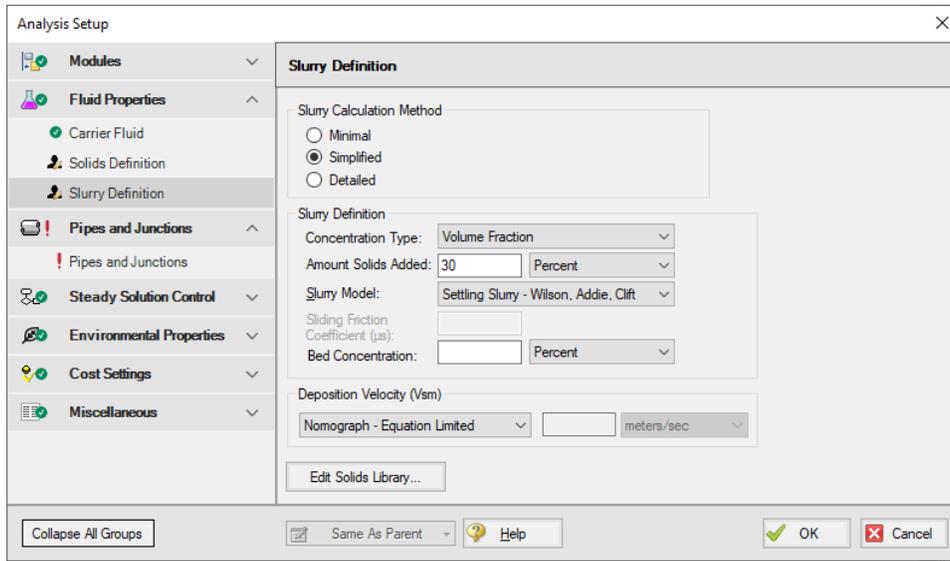


Figure 3: Slurry Definition panel in the Fluid Properties group in Analysis Setup with SSL activated

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

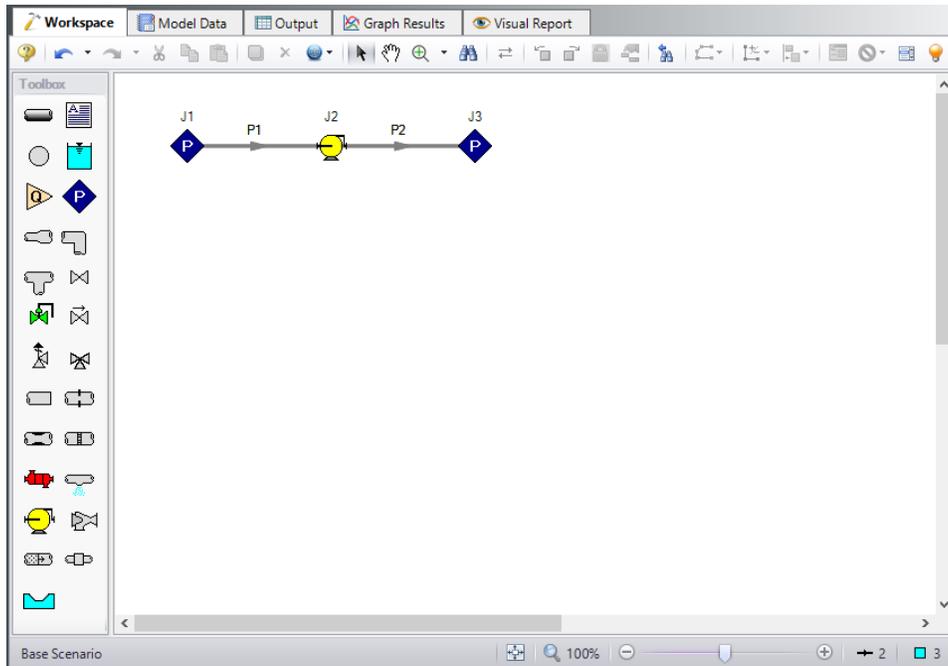


Figure 4: Layout of pipe system for Pump Sizing for Sand Transfer System example

Enter the following pipe and junction properties.

Pipe Properties

1. All Pipes
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 6 inch
 - d. Type = STD (schedule 40)
 - e. Friction Model Data Set = Standard
 - f. Length =

Pipe	Length (meters)
1	15
2	305

Junction Properties

1. J1 & J3 Assigned Pressures
 - a. Elevation = 0 meters
 - b. Pressure = 10.34 barG (1034 kPa (g))
 - c. Pressure Specification = Stagnation
2. J2 Pump
 - a. Inlet Elevation = 0 meters
 - b. Pump Model = Centrifugal (Rotodynamic)
 - c. Analysis Type = Pump Curve
 - d. Enter Curve Data =

Volumetric	Head	Efficiency
m3/hr	meters	Percent
0	60	-
115	57	45
183	53	64
251	47	75
319	40	77
387	30	74

- e. Curve Fit Order = 2

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 5. Create the Scenarios

For this example, there are four cases to be examined. Create a scenario for each of the four cases, as shown in Figure 5:

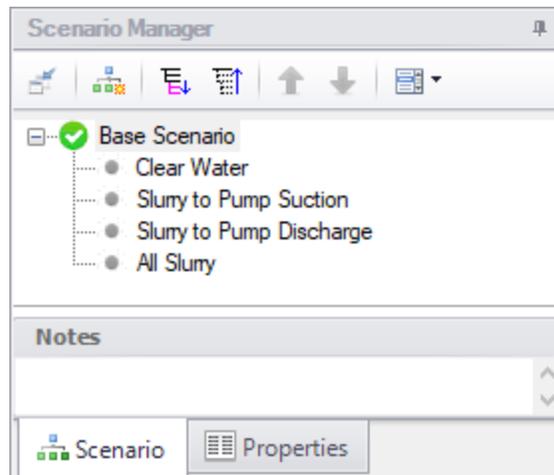


Figure 5: Use the Scenario Manager on the Quick Access Panel to create the four analysis cases

Clear Water Scenario

Load the **Clear Water** scenario by double-clicking the scenario name in the Scenario Manager.

This case requires that the pipes use clear water as their fluid. As pipes are drawn on the workspace, they automatically assume the same fluid type that is defined in the Analysis Setup window. As a result, at this point, all of the pipes are defined as using the sand slurry. Reset **both** pipes to use only water by opening the **pipe properties window** and inputting the following:

1. Fluid Properties tab
2. Fluid Properties button
3. In the Analysis Setup window that appears, click the **Solids Definition** item, and select **No Solids Added**, as shown in Figure 6. This will treat the fluid as clear water.
4. Click **OK**.
5. Make sure to do this to both pipes.

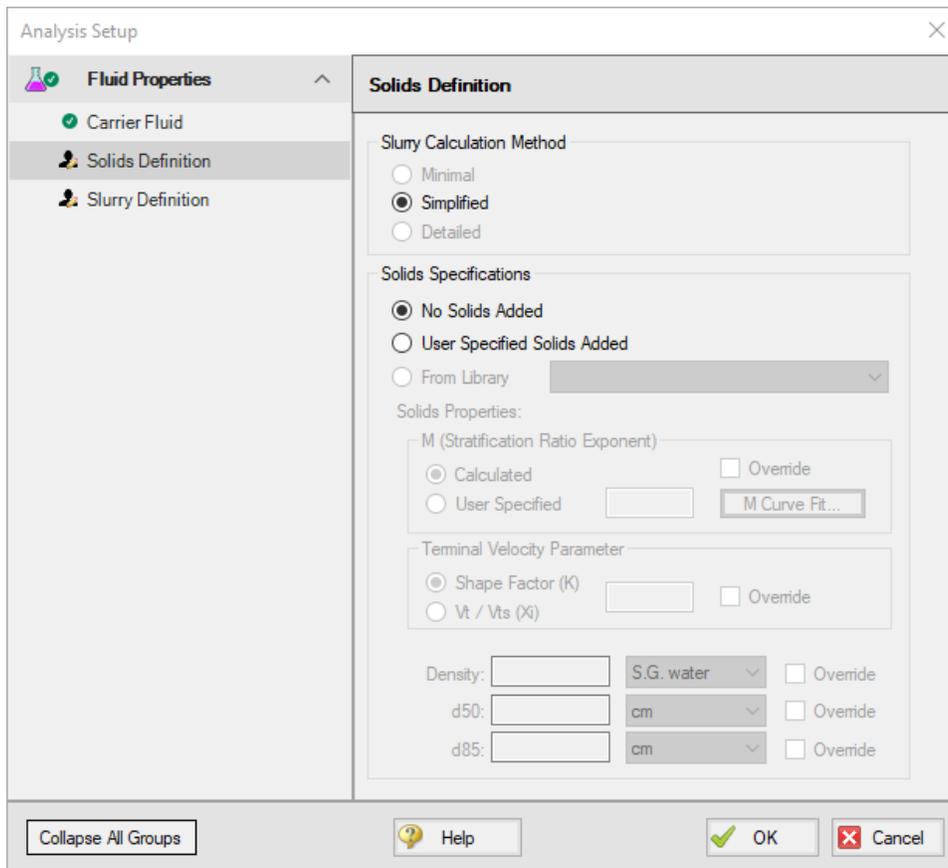


Figure 6: Fluids can be treated as clear fluid by selecting No Solids Added as the Solids Specification

Slurry to Pump Suction Scenario

Now load the **Slurry to Pump Suction** as the active scenario in the Workspace.

For the case where the fluid far upstream of the pump suction is slurry, and the fluid just upstream of the pump is clear water, a [Volume Balance junction](#) must be used to account for the difference in fluid density. The Volume Balance junction balances the flow based on volume rather than mass, ensuring that the velocity across the interface is the same. This junction must be used any time there is a system interface with fluids of different densities.

Add a Volume Balance junction just upstream of the pump suction by splitting the pipe as follows:

1. While holding the **Shift** key drag and drop the **Volume Balance junction** over pipe **P1**.
2. In the dialogue box set Pipe A as **14.99 meters**, then select **OK**. The Workspace should now appear as shown in Figure 7. This will insert the fluid interface shortly before the pump suction.
3. Open the **Volume Balance properties window** for junction J4 and check that the elevation is **0 meters**.

This case requires that the short pipe just upstream and the pipe downstream of the pump use clear water as their fluid. Set **both** pipes P2 and P3 to use only water as follows:

1. Fluid Properties tab
2. Fluid Properties button
3. In the Analysis Setup window that appears, click the **Solids Definition** item, and select **No Solids Added**, as shown in Figure 6. This will treat the fluid as clear water.
4. Click **OK**.
5. Make sure to do this to **both** pipes.

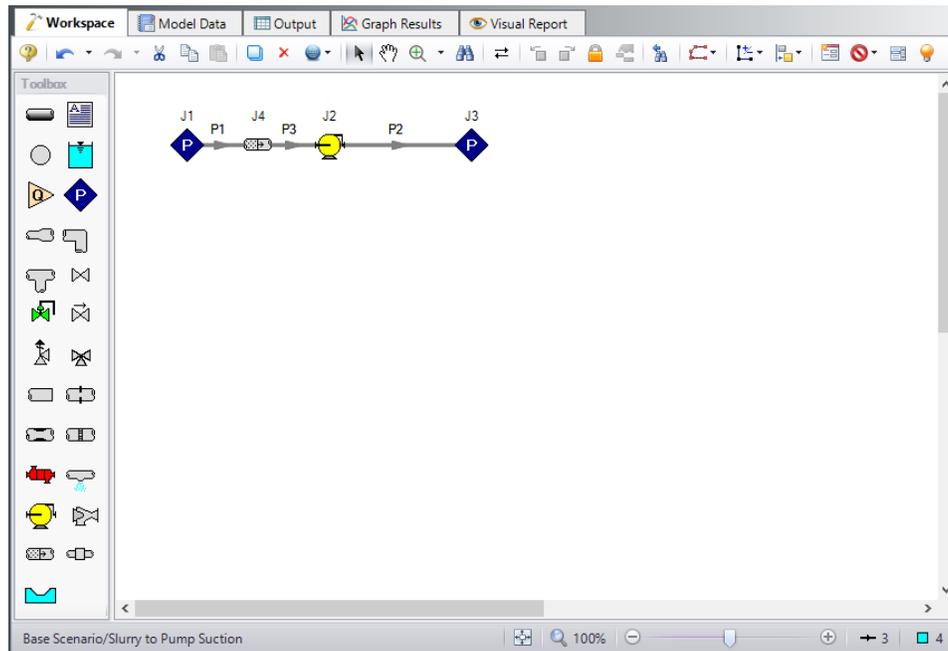


Figure 7: Layout of pipe system for partial slurry case with slurry at the pump suction showing the Volume Balance junction for the Slurries with Variable Fluid Properties example

Slurry to Pump Discharge Scenario

Load the **Slurry to Pump Discharge** scenario as the active scenario.

For the case where the fluid upstream and just downstream of the pump discharge is slurry, and the rest of the system downstream is clear water, a Volume Balance junction must be used again to account for the difference in fluid density.

Add a Volume Balance junction just downstream of the pump discharge by splitting pipe 2.

1. While holding the **Shift** key drag and drop the **Volume Balance junction** over pipe **P2**.
2. In the dialogue box set Pipe A as **0.01 meters**, then select **OK**. The Workspace should now appear as shown in Figure 8. This will insert the fluid interface shortly after the pump discharge.
3. Open the **Volume Balance properties window** for junction J5 and check that the elevation is **0 meters**.

This case requires that the pipe P4 downstream of the pump use clear water as its fluid. Set pipe **P4** to use only water as follows:

1. Fluid Properties tab
2. Fluid Properties button
3. In the Analysis Setup window that appears, click the **Solids Definition** item, and select **No Solids Added**, as shown in Figure 6. This will treat the fluid as clear water.
4. Click **OK**.

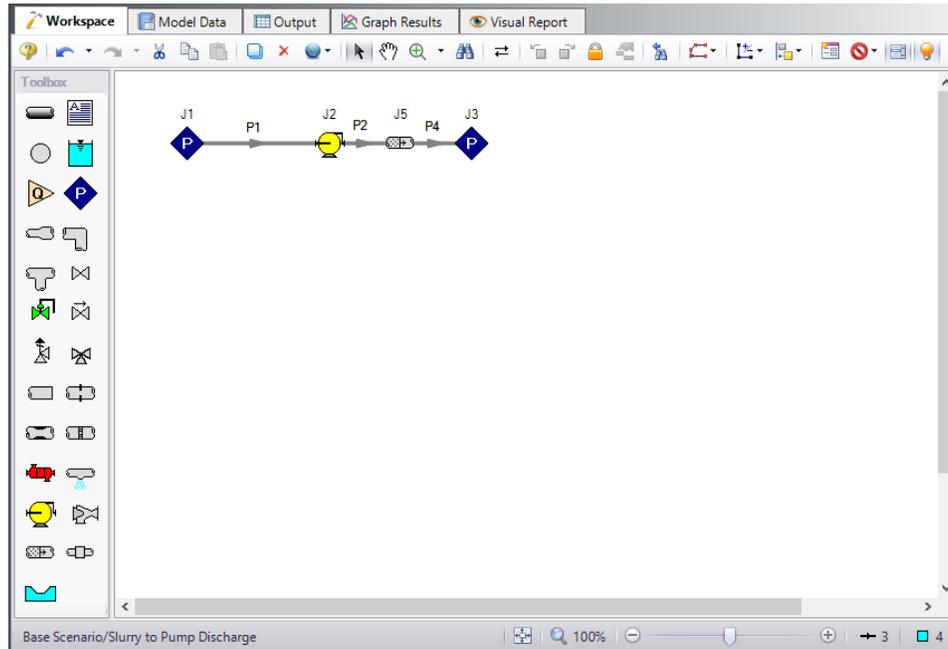


Figure 8: Layout of pipe system for partial slurry case with slurry at the pump discharge showing the Volume Balance junction for the Slurries with Variable Fluid Properties example

All Slurry Scenario

For this case, all of the pipes contain the sand slurry. The system layout is the same as is shown in Figure 4.

Step 6. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Note: All scenarios can be run automatically by Fathom by completing a Batch Run from the File menu.

Step 7. Examine the Output

Table 1 shows the variation of the system flows, velocities, and pump performance as the slurry progresses through the system.

Figure 9 shows the pump curve plotted along with both the water and slurry system curves. This graph was generated in the All Slurry scenario using the Slurry System Curve graph type.

As the slurry progresses from the system inlet to the pump suction, the system volumetric flow rate, pump head, and power are not significantly changed (compare Table 1 output for Clear Water to Slurry to Pump Suction).

After the slurry arrives at the pump discharge (compare Table 1 output for Slurry to Pump Suction to Slurry to Pump Discharge), there is a notable increase in the system volumetric flow rate (318.8 to 362.5 m³/hr). This is a result of the new density passing into the pump, which results in a different velocity in the pipe line (4.752 to 5.403 meters/sec), a different pressure loss (which varies as velocity squared, and the velocity is now higher), and hence a different head rise on the pump. This shifts the pump operation out on the curve to a higher flow rate and lower head (39.59 to 33.78 meters).

Eventually the slurry fills the entire pipe (compare Table 1 output for Slurry to Pump Discharge to All Slurry). Because of the increased overall head loss of the slurry compared to water, the system shifts back on the pump curve to a lower flow rate (362.5 to 339.6 m³/hr) and higher head rise (33.78 to 36.92 meters). The system operating velocity also drops (5.403 to 5.061 meters/sec).

The power requirement on the pump increases from the Clear Water to All Slurry scenario as the system progresses from all water to all slurry.

Of significance here is the fact that, perhaps counter to intuition, the clear water and all slurry cases do not encompass the entire flow and system performance cases for the pump. The highest and lowest flow cases actually occur as the system is only partially full of slurry. Care should be taken to examine intermediate slurry flow cases to determine system feasibility for situations where slurry is being introduced into the flow, as the maximum and minimum flow cases will not necessarily occur when the system is flowing all clear fluid, or all slurry.

It is difficult to quantify the intermediate cases through the use of pump vs. system curves. This is a result of the variation in density that occurs in the system, which confuses the meaning of Head which itself depends on constant density. Thorough analysis is needed in these cases. Figure 9 shows the clear water and all slurry system curves.

Finally, note that to keep the example clearer, the effect of pump de-rating was not addressed. In real systems when the slurry progresses from the Slurry to Pump Suction case to the Slurry to Pump Discharge case, there will be a further decrease in pump performance which will effectively lower the pump curve in Figure 9. This would serve to shift operation to even higher flow rates for the Slurry to Pump Discharge and All Slurry cases.

Table 1: System flow and pump performance for different scenarios. The velocity and volumetric flow results are for the inlet of the pipe at the pump discharge

Parameter	Vol. Flow Rate	Velocity	dP	dH	Overall Power
Units	m ³ /hr	meters/sec	bar	meters	kW
Clear Water	320.4	4.774	3.868	39.40	44.17
Slurry to Pump Suction	318.8	4.752	3.887	39.59	44.17
Slurry to Pump Discharge	362.5	5.403	4.953	33.78	65.55
All Slurry	339.6	5.061	5.413	36.92	65.94

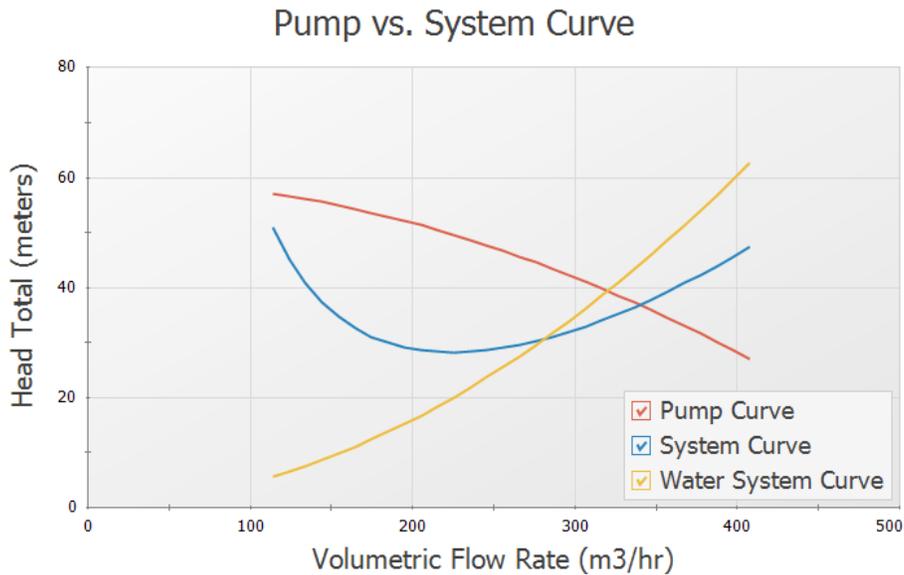


Figure 9: Plotting the pump curve along with the water and slurry system curves illustrates how the pump performance will vary as the slurry progresses through the system

Slurry System Feasibility Study - SSL (Metric Only)

Summary

Note: Because this example is presented in SI units in the reference from which it was taken, it is presented in the same units here for consistency when discussing the results.

This example demonstrates how the Settling Slurry (SSL) add-on module can be used to evaluate the feasibility of a slurry system operating point. The example is based on Case Study 13.1 found on page 329 of *Slurry Transport Using Centrifugal Pumps*, 3rd Edition, by Wilson, Addie, Sellgren, and Clift. Here only the 0.65 m case is evaluated.

An operating condition from a previous case study was selected:

Condition A:

$D = 0.65$ meters

$V_{sm} = 4.84$ meters/sec

$C_{vd} = 0.20$

$V_m = 6.30$ meters/sec

It was noted in the previous study that increasing C_{vd} and decreasing V_m reduced the specific energy consumption, and a revised operating point was also selected:

Condition B:

$D = 0.65$ meters

$V_{sm} = 4.84$ meters/sec

$C_{vd} = 0.221$

$V_m = 5.70$ meters/sec

Using these two operating points, determine whether either operating point would be feasible for the system.

Note: This example can only be run if you have a license for the SSL module.

Topics Covered

- Entering solids data
- Reviewing calculated slurry output
- Evaluating stability of system operability

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition the user should have worked through [Beginner - Pump Sizing for Sand Transfer System - SSL](#) example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Slurry System Feasibility Study - SSL.fth***
- ***Slurry System Feasibility Study - SSL.xlsx***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate SSL and select Settling to enable the SSL module for use. The items in the Fluid Properties group will change to accommodate slurry definitions. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the new option called Slurry. From here you can quickly toggle between Disable mode (normal AFT Fathom) and Settling (SSL mode).

Step 3. Define the Fluid Properties Group

Open Analysis Setup from the toolbar or from the Analysis menu and input the following on the respective panels.

1. Carrier Fluid - Figure 1
 - a. Fluid Model = Basic (Water)
 - b. Temperature = 4 deg. C
2. Solids Definition - Figure 2
 - a. Slurry Calculation Method = Detailed
 - b. Solids Specifications = User Specified Solids Added

- c. M (Stratification Ratio Exponent) = Calculated
 - d. Terminal Velocity Parameter = V_t / V_{ts} (Xi)
 - e. Value = 0.56
 - f. Density = 2.65 S.G. water
 - g. d_{50} = 0.7 mm
 - h. d_{85} = 1 mm
3. Slurry Definition - Figure 3
- a. Slurry Calculation Method = Detailed (this selection will be the same as on the Solids Definition panel)
 - b. Concentration Type = Volume Fraction
 - c. Amount Solids Added = 20 Percent

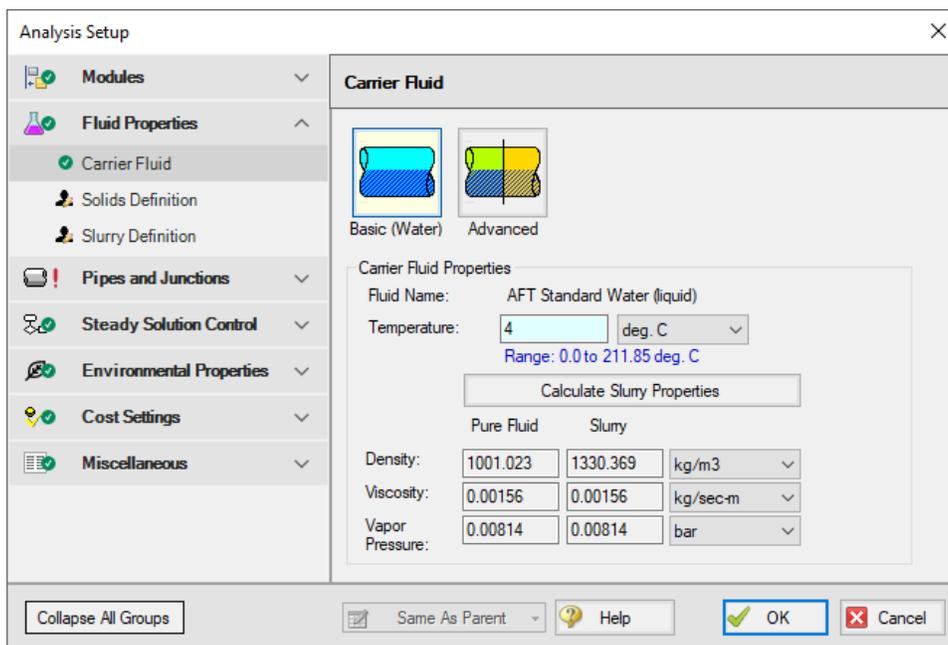


Figure 1: Carrier Fluid panel in the Fluid Properties group in Analysis Setup with SSL activated

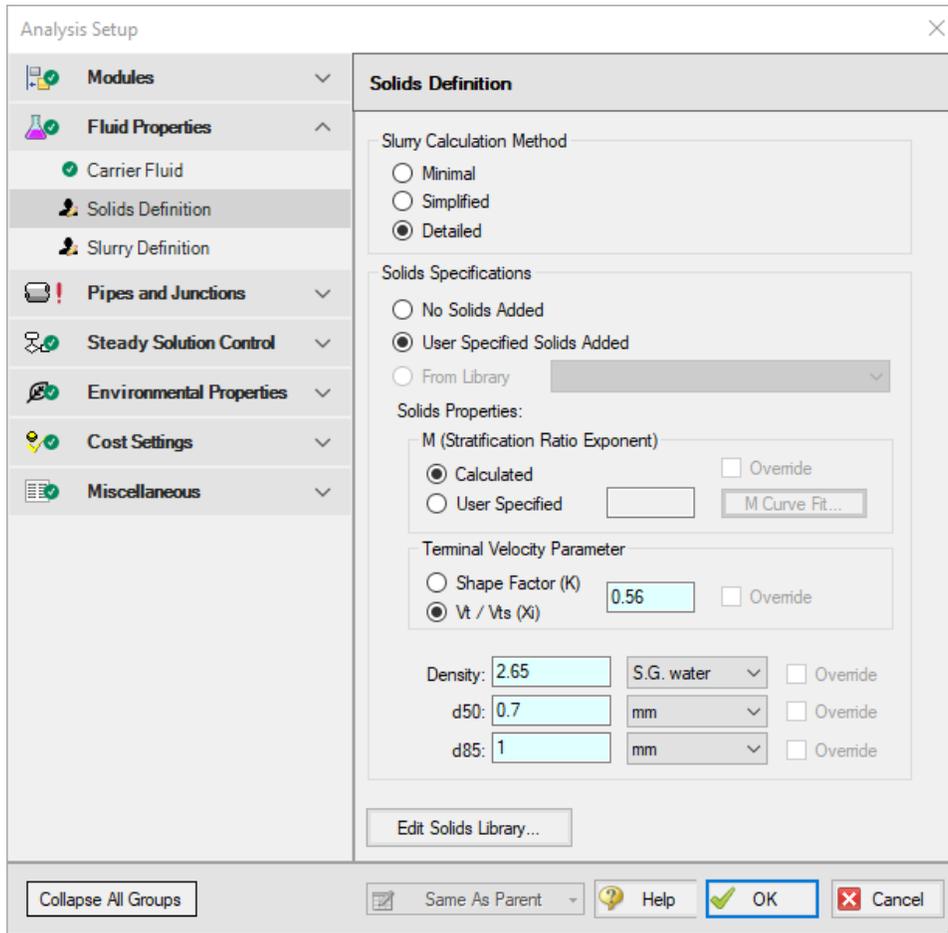


Figure 2: Solids Definition panel in the Fluid Properties group in Analysis Setup with SSL activated

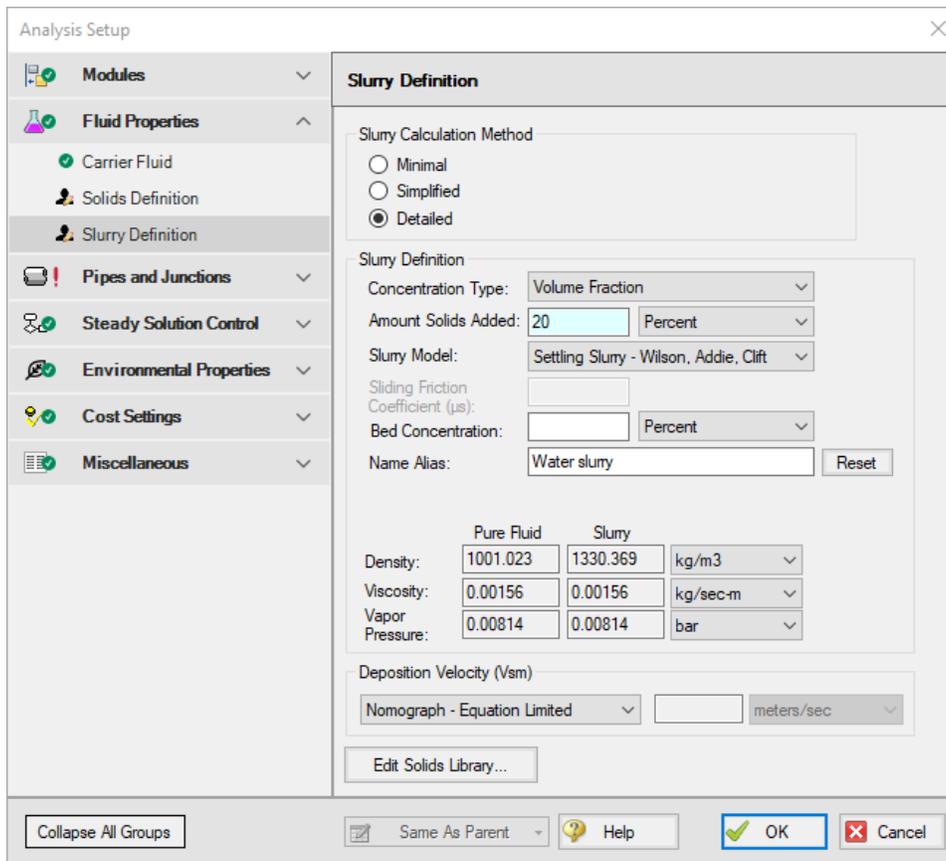


Figure 3: Slurry Definition panel in the Fluid Properties group in Analysis Setup with SSL activated

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

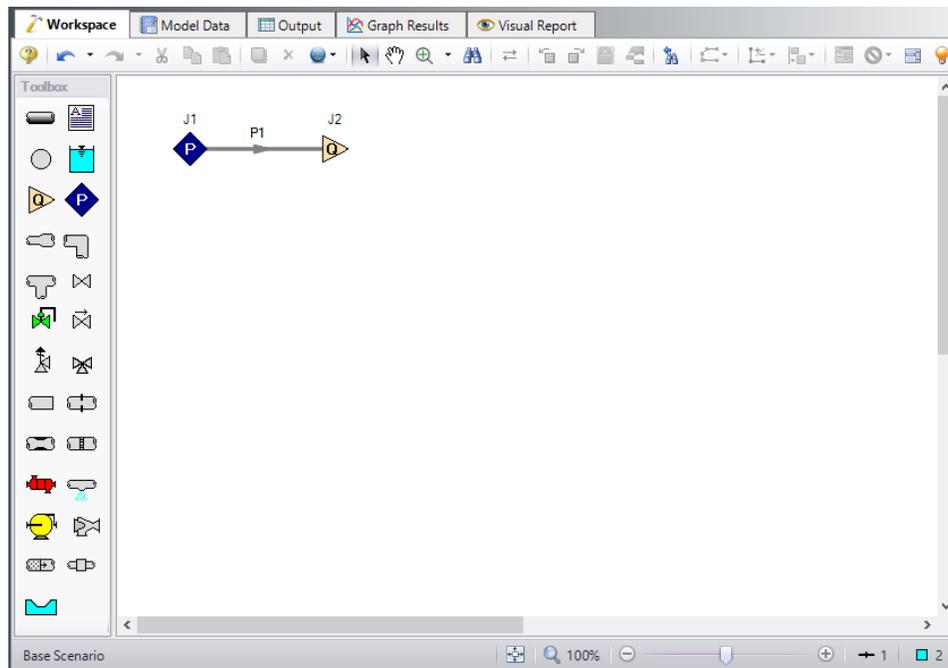


Figure 4: Layout of pipe system for Slurry System Feasibility Study example

Enter the following pipe and junction properties.

Pipe Properties

1. All Pipes
 - a. Pipe Material = (User Specified)
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Inner Diameter = 0.65 meters
 - d. Friction Model Data Set = User Specified
 - e. Friction Factor = Explicit Friction Factor
 - f. Value = 0.012
 - g. Length = 100 meters

Junction Properties

1. J1 Assigned Pressure
 - a. Elevation = 0 meters
 - b. Pressure = 10 bar
 - c. Pressure Specification = Stagnation
2. J2 Assigned Flow

- a. Elevation = 0 meters
- b. Type = Outflow
- c. Flow Specification = Mass Flow Rate
- d. Flow Rate = 4000 m-ton/hr Solids (metric tons equivalent to 4.0×10^6 kg/hr)
 - i. (metric tons equivalent to 4.0×10^6 kg/hr)

➤ **Turn on Show Object Status** from the View menu to verify if all data is entered. If so, the Pipes and Junctions group in Analysis Setup will have a check mark. If not, the uncompleted pipes or junctions will have their number shown in red. If this happens, go back to the uncompleted pipes or junctions and enter the missing data.

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Examine the Output

The maximum settling velocity for Condition A calculated by Fathom is 4.833 meters/sec, as shown in Figure 5. The value for the maximum settling velocity for Condition B can be reproduced in a similar fashion using Fathom.

Pipe	Name	Im (m/100 m)	Jm (m/100 m)	Velocity (meters/sec)	Settling Velocity Maximum (meters/sec)	Vm/Vsm	dH Slurry (meters)	dP (bar)	Mass Flow Rate (kg/sec)	Vol Flow Slurry (m3/hr)	(Im - Iw) / (Sm - Sw)
1	Pipe	6.120	4.596	6.323	4.833	1.308	4.596	0.6010	2.791	7.554	0.07152

Jct	Name	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)	Vol. Flow Rate Thru Jct (m3/hr)	Mass Flow Rate Thru Jct (kg/sec)	Loss Factor (K)
1	Assigned Pressure	9.734	9.734	10.000	10.000	7.554	2.791	0
2	Assigned Flow	9.133	9.133	9.399	9.399	7.554	2.791	0

Figure 5: Output for Condition A calculation showing maximum settling velocity

Step 7. Evaluate System Operability

To determine the feasibility of the selected operating conditions, it is helpful to examine how the operating points relate to the system operability. One method of evaluating the system operability is to plot the system curves in terms of head of slurry (Jm vs. velocity). It is useful to plot the system curves as lines of constant concentration, as well as lines of constant slurry throughput.

The system curves for a given slurry throughput and concentration can be plotted directly by using the graphing capabilities of Fathom. However, to examine the system operability over a range of flows and concentrations, it may be more convenient to use a spreadsheet application, such as Microsoft Excel, to plot the system data all together. Fathom's scenario manager and multi-scenario output features can be utilized to facilitate this process.

Evaluating the system in this manner may require a large number of scenarios to sufficiently capture the system operability. For your convenience, all of the necessary cases have been set up in the Fathom model file ***Metric - Slurry System Feasibility Study - SSL.fth***.

Steps 8 through 12 describe how the cases were set up in Fathom, and then copied into a spreadsheet for evaluation, should you wish to build the evaluation cases for yourself. The final evaluation of the operating points is presented in Step 13.

Step 8. Creating Scenarios to Evaluate System Operability

For this example, scenarios can be used to calculate Jm and velocity data for three different lines of constant concentration, as well as for three different lines of constant throughput.

Setup the scenarios for constant concentrations as follows:

1. From the Scenario Manager on the Quick Access Panel, create a scenario under the Base Scenario called *0.65 meters pipe*.
2. Under the *0.65 meters pipe* scenario, create a scenario called *Cvd = 0.15*. Load this scenario, and open the Analysis Setup window. Set the solids concentration to 15% in the Slurry Definition panel.
3. Under the *Cvd = 0.15* scenario, create scenarios for a range of mass flow rates from 1,000 m-ton per hr to 10,000 m-ton per hr in increments of 500 m-ton per hr, as shown in Figure 6. Load each of these scenarios and set the flow value in each Assigned Flow junction to the corresponding throughput value for each scenario.
4. Right-click scenario *Cvd = 0.15* and select Clone With Children. Name the new scenario *Cvd = 0.20*. In this scenario, open Analysis Setup and change the solids concentration to 20%. Repeat this process, and create a set of scenarios for *Cvd = 0.34*.
5. There are now three groups of scenarios, each for a different solids concentration. Each of the three concentrations will have a set of scenarios for a range of different through puts, as shown in Figure 7.

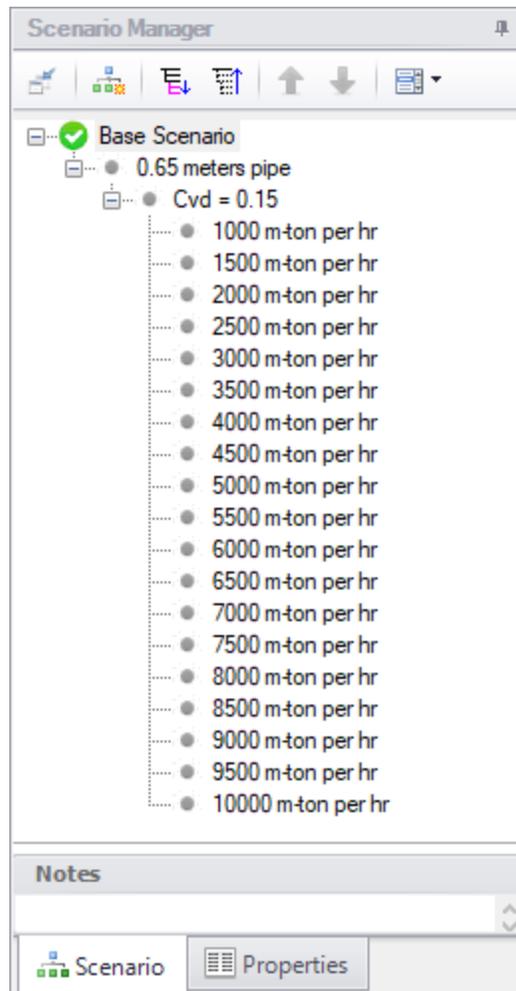


Figure 6: The Scenario Manager on the Quick Access Panel can be used to set up multiple throughput cases for a given solids concentration

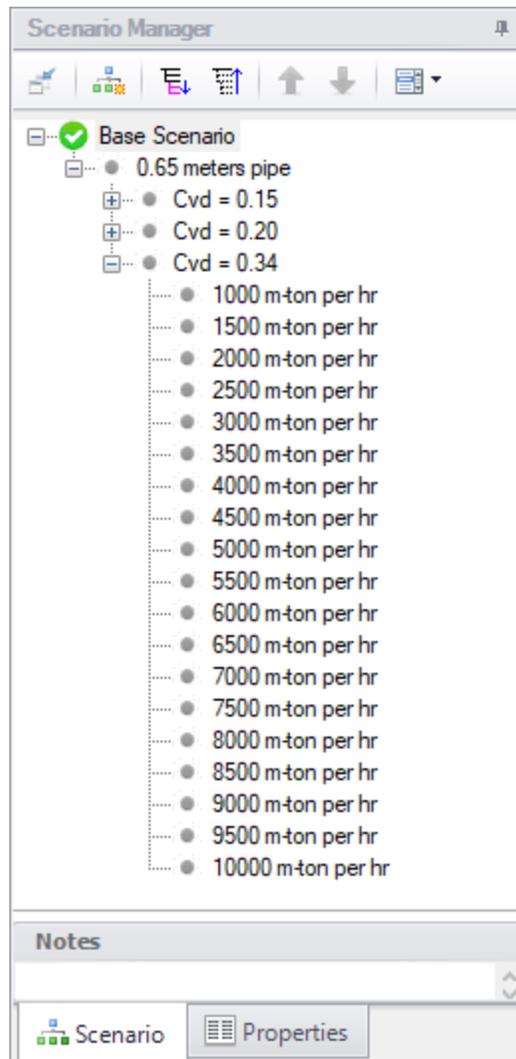


Figure 7: The Scenario Manager on the Quick Access Panel can be used to clone scenarios to easily create additional constant solids concentration cases

Setup the scenarios for constant throughput as follows:

1. Under the *0.65 meters pipe* scenario, create a scenario called *Flow = 3500 m-ton per hr*. Load this scenario, and set the flow value in the Assigned Flow junction 3500 m-ton/hr.
2. Under the *Flow = 3500 m-ton per hr* scenario, create scenarios for a range of solids concentrations, as shown in Figure 8. Load each of these scenarios and set the solids concentration value in the Slurry Definition panel in Analysis Setup to the corresponding solids concentration value for each scenario.
3. Right-click scenario *Flow = 3500 m-ton per hr* and select Clone With Children. Name the new scenario *Flow = 4000 m-ton per hr*. In this scenario, set the flow value in the Assigned Flow junction 4000 m-ton/hr. Repeat this process, and create a set of scenarios for *Flow = 4500 m-ton per hr*.

- There are now three additional sets of scenarios, each for a different throughput value. Each of the three flows will have a set of scenarios for a range of different solids concentrations, as shown in Figure 9.

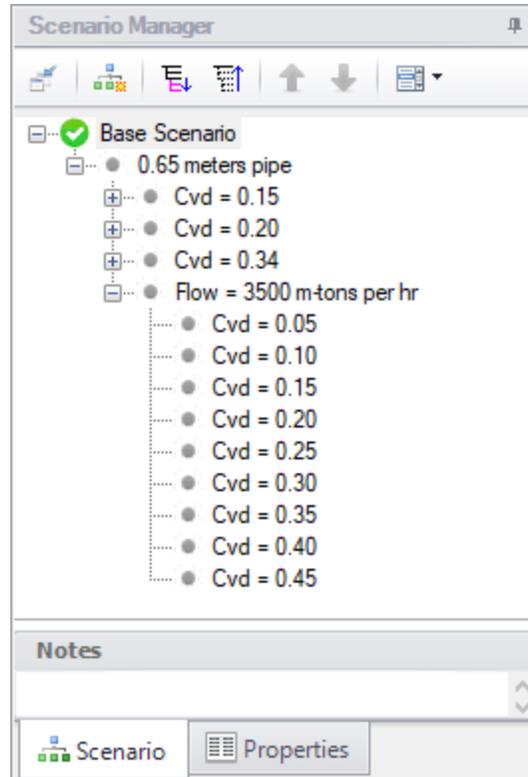


Figure 8: The Scenario Manager on the Quick Access Panel can be used to set up multiple solid concentration cases for a given throughput

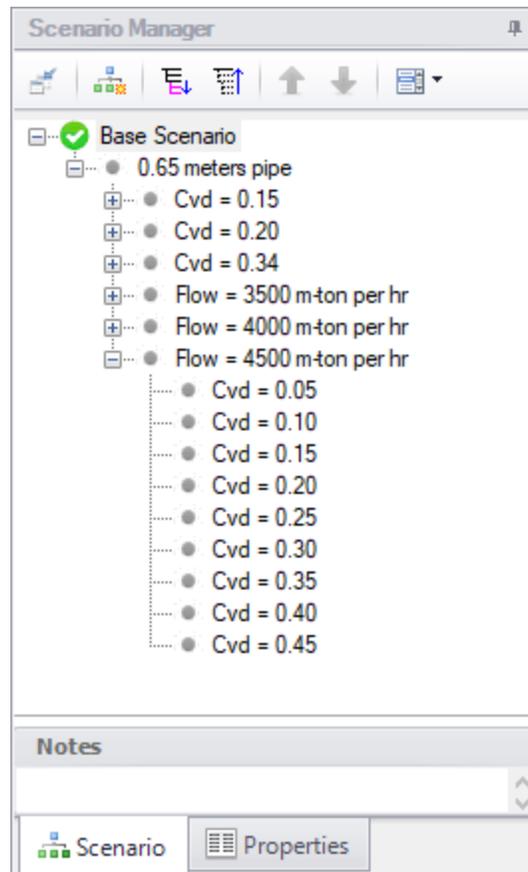


Figure 9: The Scenario Manager on the Quick Access Panel can be used to clone scenarios to easily create additional constant throughput cases

Step 9. Specify Output

Make sure that the Base Scenario is active, then open Output Control by selecting Output Control from the toolbar or Tools menu.

Knowing that for this example, the output data will be exported to a spreadsheet application, use the output control to set up the slurry output in a way that will make it simpler to plot the data in the spreadsheet. Open Output Control and on the Display Parameters tab, click the Slurry button. Then select the specific data to be displayed on the Slurry tab in the Output Window, as well as the order in which the data appears.

Figure 10 illustrates how the slurry output data could be organized in the Output Window. Because we will be plotting Jm vs. velocity in the spreadsheet, velocity and Jm are displayed next to each other in the output. One might also wish to plot Im vs. velocity, so velocity can be displayed again adjacent to the output data for Im.

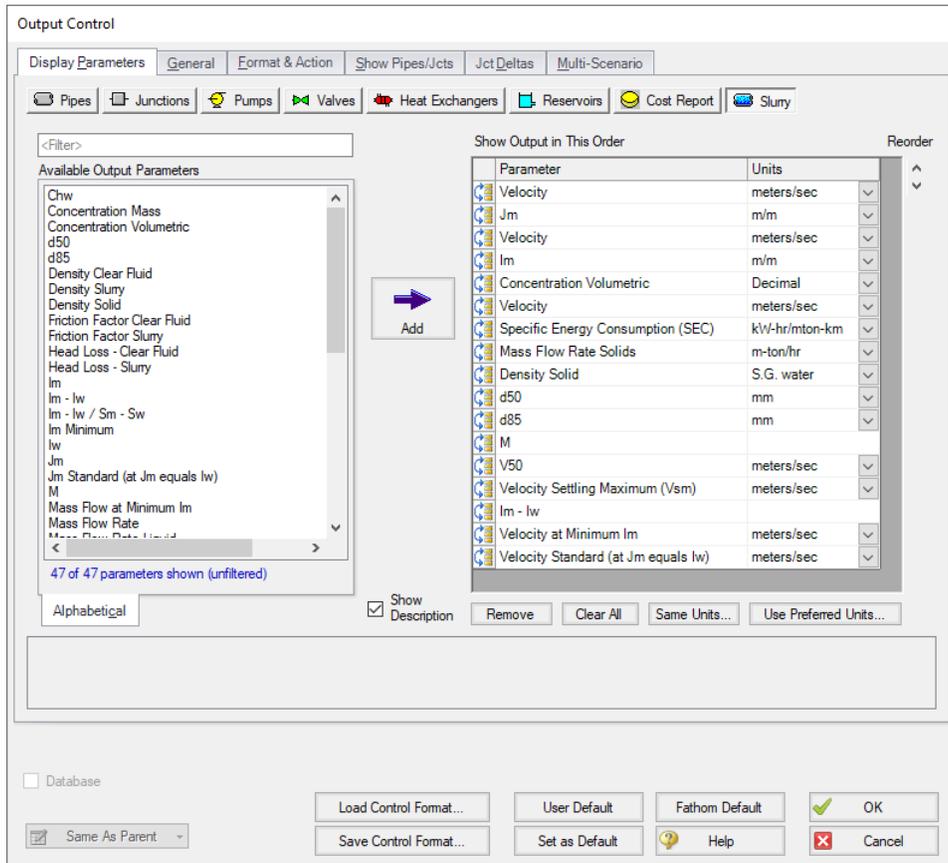


Figure 10: The Output Control window can be used to configure the slurry output into a format convenient for importing into a spreadsheet application

For this example, the multi-scenario output feature can also be used to simplify exporting the output data to a spreadsheet application. This feature allows the output from multiple scenarios to be viewed simultaneously. Load the *1000 m-ton per hr* scenario that is under the $Cvd = 0.15$ scenario. Open the **Output Control** window and select the **Multi-Scenario** tab. Under Display Type, click **Selected Scenarios** and the Select Scenarios window appears. Check the box beside each scenario for which data is to be displayed in the Output, as shown in Figure 11. 84 of 92 scenarios should be selected. Click **OK** when finished.

Under Scenario Name Format, select **Show Scenario Ancestral Name**. Under Display Current Scenario, select **In the Order Listed**, if not selected already. Click the **Clear All Colors** button to make all scenario rows white.

This information should now appear, as shown in Figure 12. Close the Output Control window by clicking **OK**.

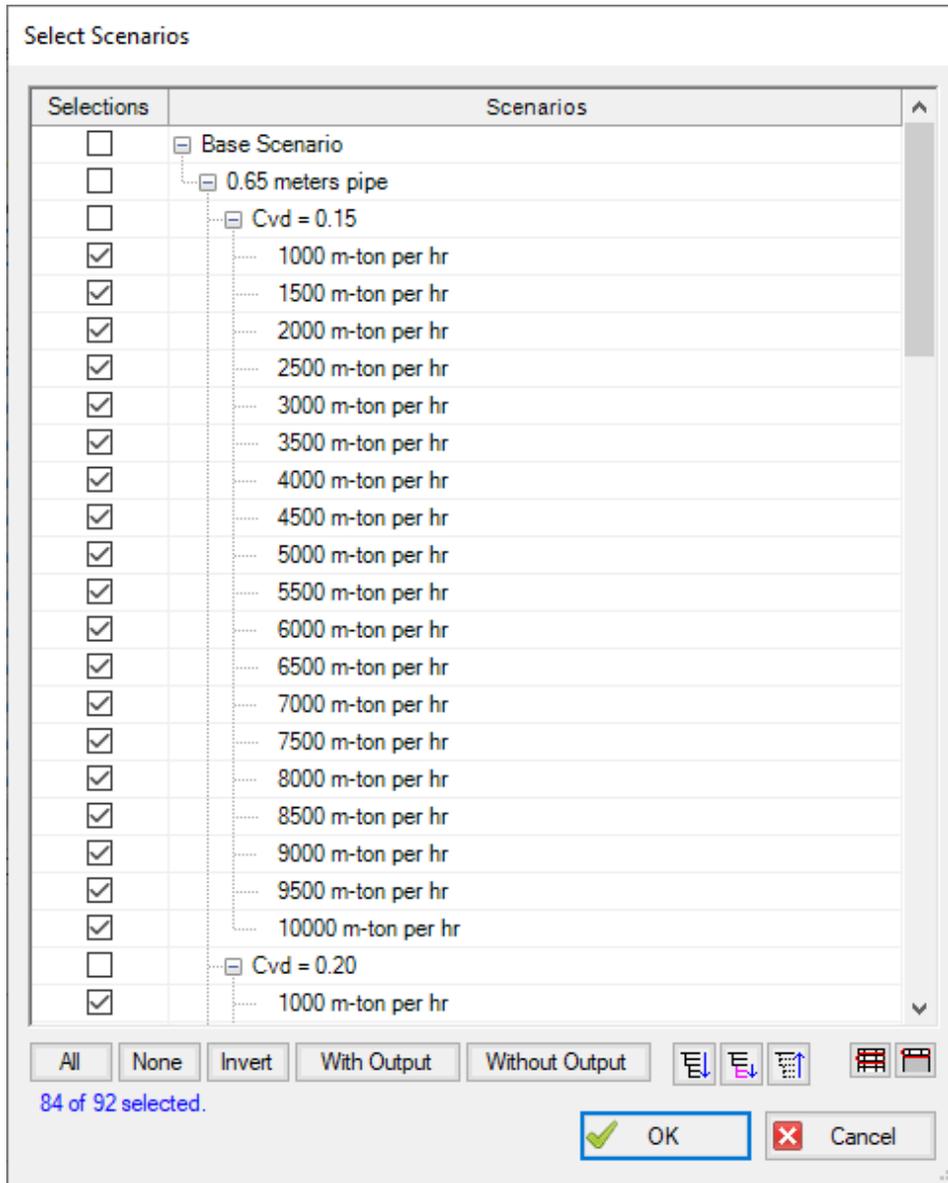


Figure 11: Multiple scenarios can be selected for display in the Output window

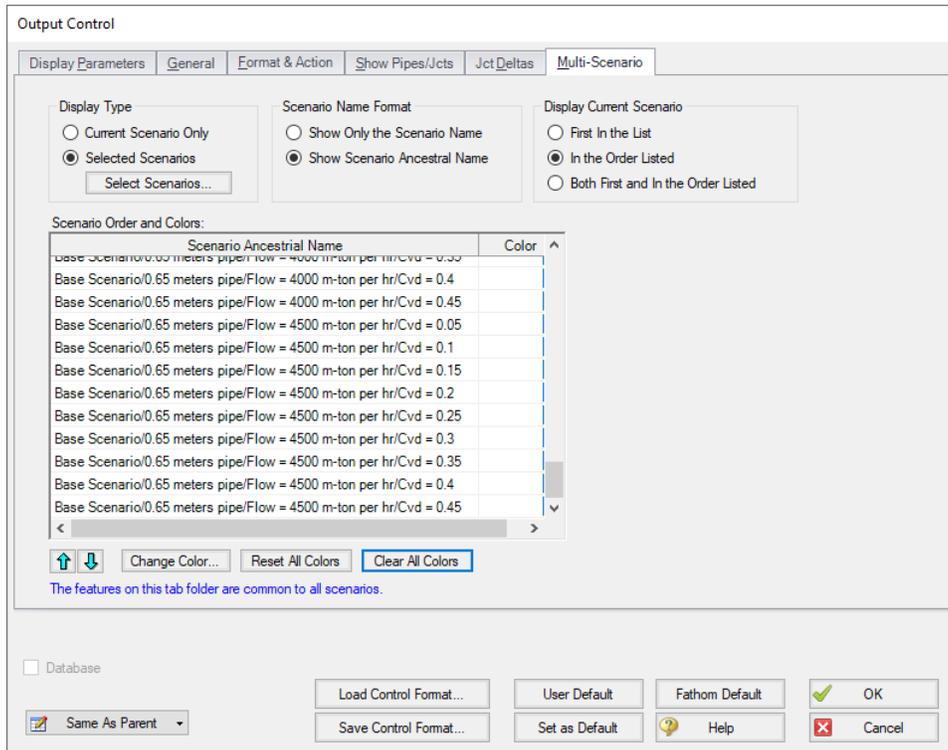


Figure 12: The Output Control window can be used to display the output from multiple scenarios simultaneously

Step 10. Run the Model

All of the different cases that have been set up for this example in each of the scenarios must now be run so the results can be examined. Because there are so many scenarios, the Batch Run feature in the File menu can be used to make running all of the different scenarios much easier.

From the File menu, select Start Batch Run. Under Batch Run Type, Scenarios In Current Model should be selected. Click Add Scenarios, and in the Select Scenarios window, select all 84 of the scenarios to be run, then click OK. All of the selected scenarios will be displayed in the Scenarios to Run list, as shown in Figure 13.

Click Start Run to start the analysis. The batch run manager will load and run each of the listed scenarios sequentially.

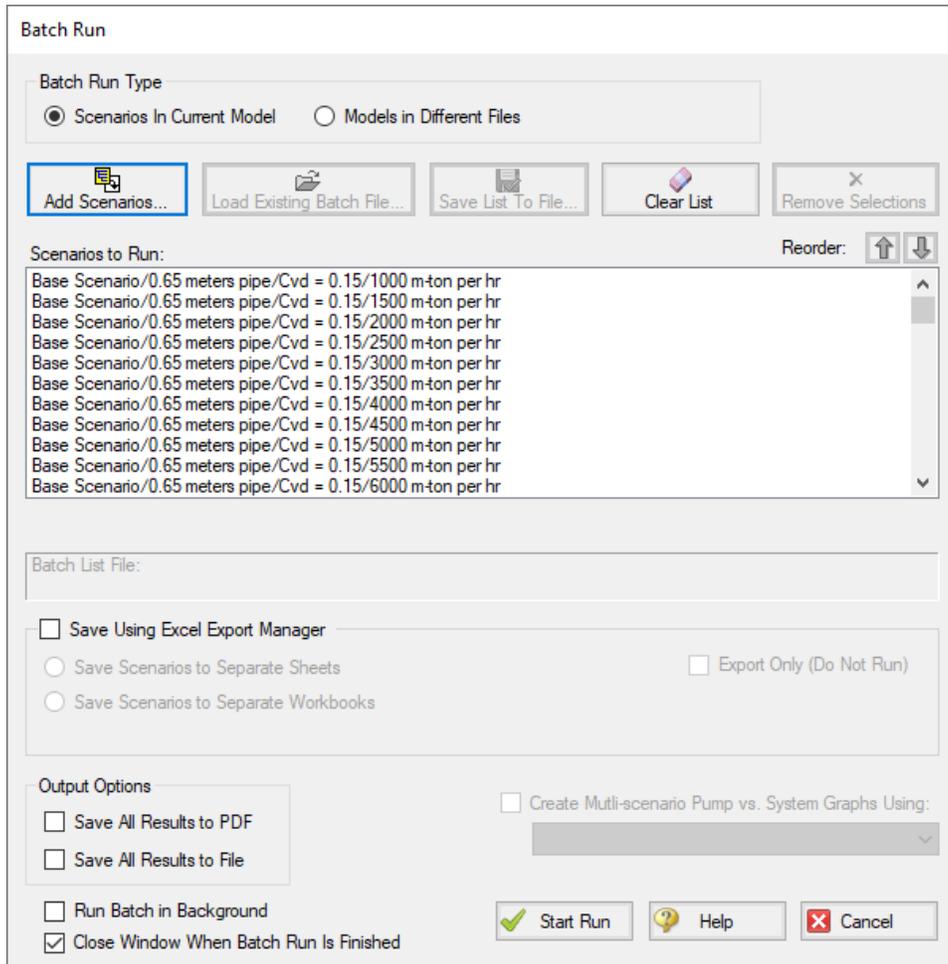


Figure 13: The Batch Run Manager is used to automatically run multiple scenarios

Step 11. Examine the Output

Once the batch manager has finished running all of the selected scenarios, load the scenario in which the multi-scenario output was set up (1000 m-ton per hr scenario that is under the Cvd = 0.15 scenario).

The Output window contains all the data that was specified in the Output Control window. Because the multi-scenario output feature was selected, it will also display the calculated output for each of the selected scenarios simultaneously. Figure 14 shows the multi-scenario output on the Slurry tab.

Slurry System Feasibility Study - SSL

Pipe	Slurry	Velocity (meters/sec)	Jm (m/s)	Velocity (meters/sec)	Im (m/s)	C _s (Decimals)	Velocity (meters/sec)	SEC (h/h-hr-ton per hr)	Mass Flow Solids (m-ton/hr)	Density Solid (g/cm ³)	d50 (mm)	d85 (mm)	M	ISO (meters/sec)	Setting Velocity Maximum (meters/sec)	Im/hr	V at Min (meters/sec)	V Standard (meters/sec)
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/1000 m-ton per hr	2.1077	0.0948	2.1077	0.11864	0.15000	2.1077	0.8125	1.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.114550	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/1500 m-ton per hr	3.1616	0.0532	3.1616	0.06685	0.15000	3.1616	0.4579	1.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0574486	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/2000 m-ton per hr	4.2154	0.04159	4.2154	0.05195	0.15000	4.2154	0.3058	2.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0352277	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/2500 m-ton per hr	5.2693	0.04022	5.2693	0.05023	0.15000	5.2693	0.3440	2.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0241966	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/3000 m-ton per hr	6.3232	0.04427	6.3232	0.05530	0.15000	6.3232	0.3787	3.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0176919	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/3500 m-ton per hr	7.3770	0.05189	7.3770	0.06481	0.15000	7.3770	0.4439	3.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0149596	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/4000 m-ton per hr	8.4308	0.05223	8.4308	0.07173	0.15000	8.4308	0.5223	4.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0104926	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/4500 m-ton per hr	9.4847	0.07487	9.4847	0.09352	0.15000	9.4847	0.6405	4.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0087551	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/5000 m-ton per hr	10.5386	0.08960	10.5386	0.11192	0.15000	10.5386	0.7665	5.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0074197	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/5500 m-ton per hr	11.5925	0.10629	11.5925	0.13276	0.15000	11.5925	0.9092	5.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0062086	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/6000 m-ton per hr	12.6463	0.12481	12.6463	0.15693	0.15000	12.6463	1.0679	6.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0054422	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/6500 m-ton per hr	13.7002	0.14520	13.7002	0.18136	0.15000	13.7002	1.2420	6.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0047499	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/7000 m-ton per hr	14.7540	0.16733	14.7540	0.20902	0.15000	14.7540	1.4314	7.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0041876	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/7500 m-ton per hr	15.8079	0.19123	15.8079	0.23866	0.15000	15.8079	1.6368	7.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0037242	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/8000 m-ton per hr	16.8617	0.21685	16.8617	0.27087	0.15000	16.8617	1.8550	8.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0033372	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/8500 m-ton per hr	17.9156	0.24420	17.9156	0.30603	0.15000	17.9156	2.0890	8.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0030104	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/9000 m-ton per hr	18.9695	0.27328	18.9695	0.34133	0.15000	18.9695	2.3375	9.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0027396	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/9500 m-ton per hr	20.0233	0.30402	20.0233	0.37975	0.15000	20.0233	2.6007	9.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0024917	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.15/10000 m-ton per hr	21.0772	0.33648	21.0772	0.42030	0.15000	21.0772	2.8784	10.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0022837	4.934	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/1000 m-ton per hr	1.5808	0.18877	1.5808	0.20122	0.20000	1.5808	1.2903	1.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.2488864	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/1500 m-ton per hr	2.3712	0.09778	2.3712	0.10021	0.20000	2.3712	0.6698	1.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.1284190	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/2000 m-ton per hr	3.1616	0.06489	3.1616	0.06860	0.20000	3.1616	0.4417	2.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0765882	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/2500 m-ton per hr	3.9520	0.05040	3.9520	0.05711	0.20000	3.9520	0.3447	2.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0524170	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/3000 m-ton per hr	4.7424	0.04477	4.7424	0.05661	0.20000	4.7424	0.3262	3.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0384471	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/3500 m-ton per hr	5.5328	0.04385	5.5328	0.05839	0.20000	5.5328	0.2999	3.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0295938	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/4000 m-ton per hr	6.3232	0.04596	6.3232	0.06120	0.20000	6.3232	0.3143	4.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0237599	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/4500 m-ton per hr	7.1136	0.05025	7.1136	0.06691	0.20000	7.1136	0.3437	4.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0192979	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/5000 m-ton per hr	7.9040	0.05627	7.9040	0.07482	0.20000	7.9040	0.3848	5.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0161332	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/5500 m-ton per hr	8.6943	0.06372	8.6943	0.08485	0.20000	8.6943	0.4358	5.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0137200	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/6000 m-ton per hr	9.4847	0.07246	9.4847	0.09643	0.20000	9.4847	0.4956	6.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0118326	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/6500 m-ton per hr	10.2751	0.08237	10.2751	0.10967	0.20000	10.2751	0.5633	6.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0102260	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/7000 m-ton per hr	11.0655	0.09337	11.0655	0.12432	0.20000	11.0655	0.6386	7.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0091055	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/7500 m-ton per hr	11.8559	0.10542	11.8559	0.14036	0.20000	11.8559	0.7209	7.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0080978	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/8000 m-ton per hr	12.6463	0.11847	12.6463	0.15774	0.20000	12.6463	0.8102	8.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0072653	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/8500 m-ton per hr	13.4367	0.13251	13.4367	0.17643	0.20000	13.4367	0.9062	8.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0065457	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/9000 m-ton per hr	14.2271	0.14750	14.2271	0.19640	0.20000	14.2271	1.0088	9.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0059396	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/9500 m-ton per hr	15.0175	0.16344	15.0175	0.21763	0.20000	15.0175	1.1178	9.500	2.650	0.7000	1.000	1.700	3.265	4.833	0.0054180	5.333	7.522
1 - Base Scenario	0.65 meters pipe/Cvd = 0.20/10000 m-ton per hr	15.8079	0.18034	15.8079	0.24010	0.20000	15.8079	1.2332	10.000	2.650	0.7000	1.000	1.700	3.265	4.833	0.0049656	5.333	7.522

Figure 14: The Output window shows the Slurry results table in the center Pipes section for multiple scenarios

Step 12. Transfer the Output Data to a Spreadsheet

The information displayed on the Slurry tab in the Output window can be copied and pasted directly into a spreadsheet application, such as Excel, as shown in Figure 15. If the scenarios and the output were set up carefully, the data will already be formatted in such a way that graphing the information using the spreadsheet graphing features can be extremely straightforward.

Using the data in the spreadsheet, plot the Jm vs. velocity data for the three lines of constant solids concentration, and the three lines of constant solids throughput, as shown in Figure 16.

Slurry System Feasibility Study - SSL

Pipe	Velocity (meters/sec)	Im (m/m)	Velocity (meters/sec)	Im (m/m)	Cv (Decimal)	Velocity (meters/sec)
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/1000 m-ton per hr	2.1077	0.095	2.1077	0.1186	0.15	2.1077
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/1500 m-ton per hr	3.1616	0.0535	3.1616	0.0669	0.15	3.1616
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/2000 m-ton per hr	4.2154	0.0416	4.2154	0.052	0.15	4.2154
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/2500 m-ton per hr	5.2693	0.0402	5.2693	0.0502	0.15	5.2693
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/3000 m-ton per hr	6.3232	0.0443	6.3232	0.0553	0.15	6.3232
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/3500 m-ton per hr	7.377	0.0519	7.377	0.0648	0.15	7.377
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/4000 m-ton per hr	8.4309	0.0622	8.4309	0.0777	0.15	8.4309
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/4500 m-ton per hr	9.4847	0.0749	9.4847	0.0935	0.15	9.4847
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/5000 m-ton per hr	10.5386	0.0896	10.5386	0.1119	0.15	10.5386
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/5500 m-ton per hr	11.5925	0.1063	11.5925	0.1328	0.15	11.5925
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/6000 m-ton per hr	12.6463	0.1248	12.6463	0.1559	0.15	12.6463
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/6500 m-ton per hr	13.7002	0.1452	13.7002	0.1814	0.15	13.7002
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/7000 m-ton per hr	14.754	0.1673	14.754	0.209	0.15	14.754
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/7500 m-ton per hr	15.8079	0.1912	15.8079	0.2389	0.15	15.8079
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/8000 m-ton per hr	16.8617	0.2169	16.8617	0.2709	0.15	16.8617
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/8500 m-ton per hr	17.9156	0.2442	17.9156	0.305	0.15	17.9156
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/9000 m-ton per hr	18.9695	0.2733	18.9695	0.3413	0.15	18.9695
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/9500 m-ton per hr	20.0233	0.304	20.0233	0.3798	0.15	20.0233
1 - Base Scenario/0.65 meters pipe/Cvd = 0.15/10000 m-ton per hr	21.0772	0.3365	21.0772	0.4203	0.15	21.0772
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/1000 m-ton per hr	1.5808	0.1887	1.5808	0.2512	0.2	1.5808
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/1500 m-ton per hr	2.3712	0.0978	2.3712	0.1302	0.2	2.3712
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/2000 m-ton per hr	3.1616	0.0646	3.1616	0.086	0.2	3.1616
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/2500 m-ton per hr	3.952	0.0504	3.952	0.0671	0.2	3.952
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/3000 m-ton per hr	4.7424	0.0448	4.7424	0.0596	0.2	4.7424
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/3500 m-ton per hr	5.5328	0.0439	5.5328	0.0584	0.2	5.5328
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/4000 m-ton per hr	6.3232	0.046	6.3232	0.0612	0.2	6.3232
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/4500 m-ton per hr	7.1135	0.0503	7.1135	0.0669	0.2	7.1135
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/5000 m-ton per hr	7.9039	0.0563	7.9039	0.0749	0.2	7.9039
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/5500 m-ton per hr	8.6943	0.0637	8.6943	0.0849	0.2	8.6943
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/6000 m-ton per hr	9.4847	0.0725	9.4847	0.0965	0.2	9.4847
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/6500 m-ton per hr	10.2751	0.0824	10.2751	0.1097	0.2	10.2751
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/7000 m-ton per hr	11.0655	0.0934	11.0655	0.1243	0.2	11.0655
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/7500 m-ton per hr	11.8559	0.1054	11.8559	0.1404	0.2	11.8559
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/8000 m-ton per hr	12.6463	0.1185	12.6463	0.1577	0.2	12.6463
1 - Base Scenario/0.65 meters pipe/Cvd = 0.20/8500 m-ton per hr	13.4367	0.1325	13.4367	0.1764	0.2	13.4367

Figure 15: The Slurry results table in the Output window can be copied and pasted into a spreadsheet application in order to utilize the graphing features of the spreadsheet to generate the slurry system curves

Step 13. Analyze the Results

The results are shown in Figure 16. The solid lines refer to the constant solids concentration cases, and the broken lines refer to the constant throughput cases.

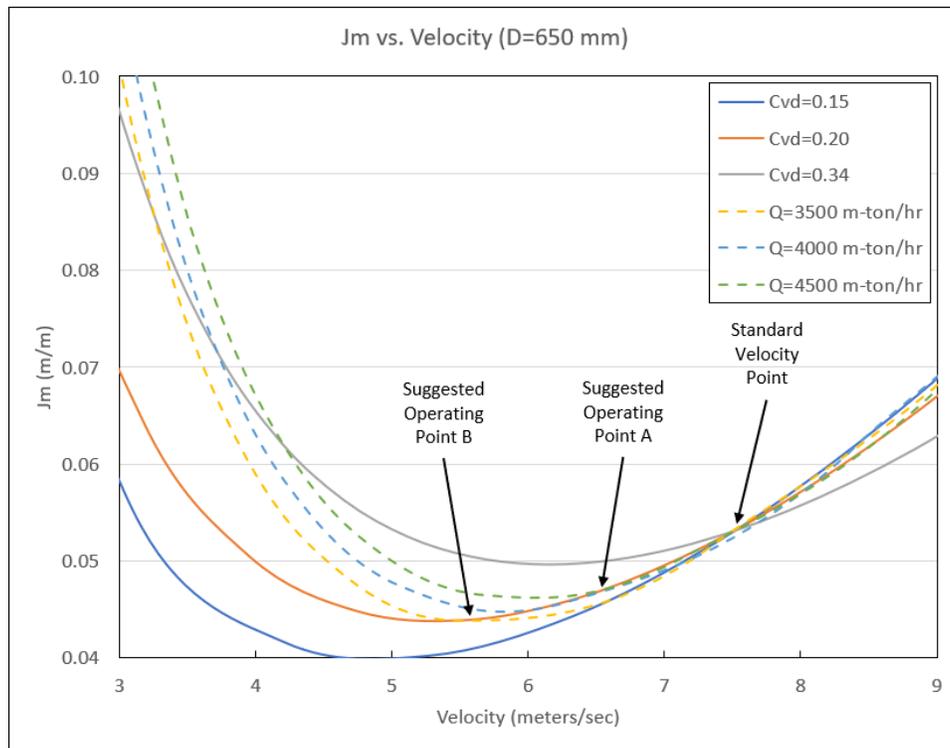


Figure 16: Slurry system curves for Slurry System Feasibility Study example

The arrows on the graph indicate the proposed operating conditions for the system on the 4000 m-ton/hr constant throughput line.

It is evident from the system characteristics that the proposed operating point Condition B, with the lower specific energy consumption, does not represent a feasible operating point for this slurry system. The system characteristics show the J_m decreasing as the mean velocity increases, so the operation would be unstable.

Even the first proposed operating condition, Condition A, is marginal. If the pump has a variable speed drive, and the solids concentration of the system does not fluctuate very much, this operating point could be used. However, as Figure 16 shows, if the concentrations vary from 3500 m-ton/hr to 4500 m-ton/hr, then the system operation is close to unstable for Condition A as well.

In order to allow for fluctuations in solids concentrations during system operations, and still maintain stable operation, it would be better to operate closer to the standard velocity. The standard velocity point is also shown in Figure 14 at about 7.5 m/s.

For this case, if operability is also considered, lower slurry concentrations, and higher energy consumption should be used to ensure system stability. Further, as discussed in the original reference, a different diameter pipe can be considered. The original reference suggests 0.55 m pipe as a better option than 0.65 m pipe because of operation closer to the standard velocity and lower capital cost.

Automated Network Sizing (ANS) Examples

The examples below utilize the Automated Network Sizing module.

Fathom ANS Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
Beginner - Three-Reservoir Model	Beginner	Water	3	0	Learn the basics of the ANS module such as defining all panels in the Sizing window to achieve a minimum flow volume objective
Control Valve	Beginner	Water	2	0	Learn how to use the ANS module and how Common Size Groups work to minimize the pipe weight of a system.
Cooling System, Creating Libraries	Intermediate	Water	21	2	Learn how to build cost libraries and engineering libraries for use with the ANS module. See how to use Scale Tables for defining pump or bend costs, how to specify material versus installation costs, and how to define costs for pipes.
Cooling System	Intermediate	Water	21	2	Learn how to use the ANS module to minimize monetary cost based on initial cost and recurring costs. See how to use the Assign Cost Libraries panel and the Sizing Summary panel. Explore how the system life and the sizing objective impacts the optimal pipe sizes.
Cooling System, Multiple Design Cases	Intermediate	Water	21	2	Learn how to use the ANS module to define and size a system based on a dependent design configuration, called a Dependent Design Case.
Hot Water System	Intermediate	Water	24	3	Learn how to use the ANS module to define a system with multiple specific design

Example	Complexity	Fluid	Pipes	Pumps	Description
					requirements and minimize initial and recurring costs. See how to use a Dependent Design Case to see how a system configuration affects the system sizing.
Variable Drive	Intermediate	Lube Oil	23	2	Learn how to use the ANS module to see the life cycle cost savings in a sized system using a Variable Frequency Drive instead of a Control Valve.
Housing Project	Advanced	Water	25	0	Learn how to use the ANS module to size a system with more than one Dependent Design Case with unique design requirements for each dependent case.

Beginner - Three Reservoir Problem - ANS

Summary

The objective of this example is to familiarize the user with the panels in the ANS module. We will apply automated sizing to determine the ideal pipe sizes from our previously built three-reservoir model to reduce costs while meeting certain requirements.

Note: This example can only be run if you have a license for the ANS module.

Topics Covered

- Minimizing the flow volume to minimize cost
- Specifying Pipe Design Requirements
- Choosing Candidate Sets

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also

watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Beginner - Three Reservoir.fth***
- ***Metric - Beginner - Three Reservoir - ANS.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the ***Metric - Beginner - Three Reservoir Problem.fth*** example file and save it as a new file.

We will be using the Base Scenario to compare the original results without Sizing to the results with Sizing. Create a child scenario named *Sized*.

The workspace should look like Figure 1 below:

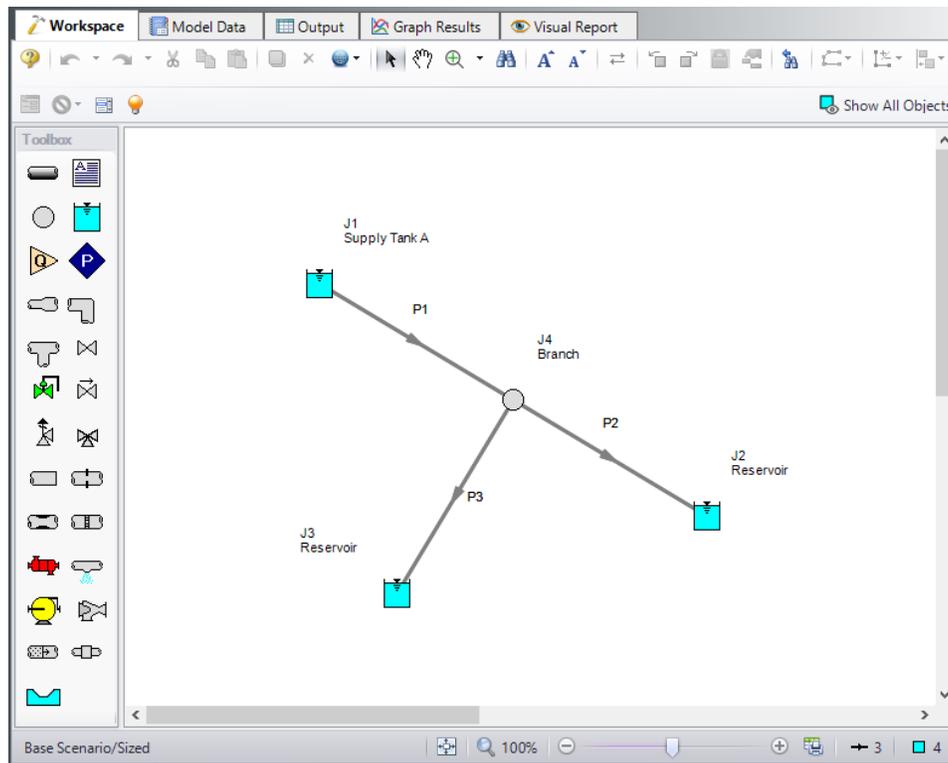


Figure 1: Layout of Three Reservoir Model in Workspace

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate ANS and select Network to enable the ANS module for use. A new group will appear in Analysis Setup titled Automatic Sizing. Click OK to save the changes and exit Analysis Setup. A new Primary Window tab will appear between Workspace and Model Data titled Sizing. Open the Analysis menu to see the new option called Automatic Sizing. From here you can quickly toggle between Not Used mode (normal AFT Fathom) and Network (ANS mode).

Figure 2 below shows the Analysis menu options when ANS is activated.

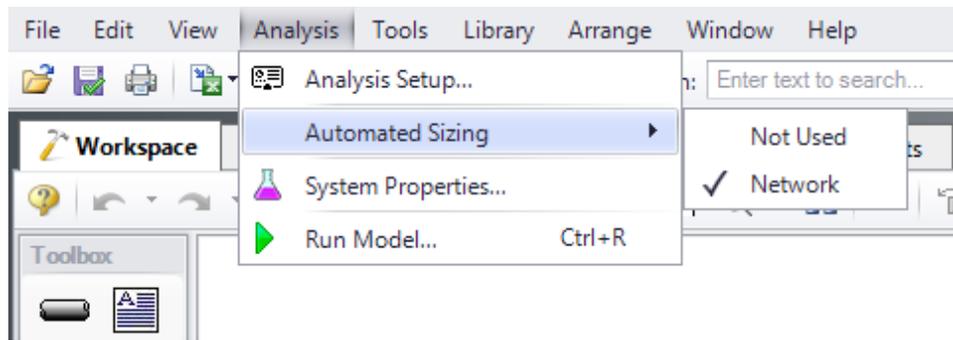


Figure 2: Activating Network Sizing from the Analysis menu

Step 4. Configure Sizing Settings

Here we will pursue a brief introduction to some of the sizing capabilities in the ANS module. A more complete discussion is given in the AFT Fathom Help file.

The Sizing window is comprised of multiple panels which can be accessed using the buttons on the Sizing Navigation panel along the bottom of the window, as shown in Figure 3. The sizing panels can be accessed in any order, though it is easiest to enter the information by navigating the panels from left to right, since the input on panels such as the Sizing Objective and Sizing Assignments panels will affect the options available on later panels. Depending on the type of sizing being done, some panels may be disabled or unused.

Each panel button contains either a green checkmark or red circle which denotes the completion status of that panel. If the minimum required information is present to run the model, the symbol will be green, whereas the red symbol represents incomplete input. The amount of information required will vary based on whether the Sizing Level Status (located on the Sizing Objective panel) is set to Perform Sizing, Calculate Costs, or Do Not Size. A detailed summary of the items which have been completed and the items which are still incomplete can be seen in the Sizing Status panel, opened from the Sizing Toolbar.

Note that some panels will always be shown as complete since the model can be run without any additional information entered on them, such as with the Design Requirements panel. However, in order to find the best system design it will often be necessary to enter more than the minimum information required by the solver.

The Lock Panel toggle located on the Sizing Toolbar prevents changes to the current panel when it is enabled. This is primarily useful to prevent editing once a scenario has been run, since any changes that are made to a scenario which has output will cause all output to be erased. By default, the ANS module will lock all panels after a sizing run is completed, requiring panels to be unlocked before any changes can be made. This setting can be changed on the Sizing Objective panel. All panels can be locked or unlocked simultaneously by using the Lock/Unlock All Panels buttons on the Sizing Toolbar.

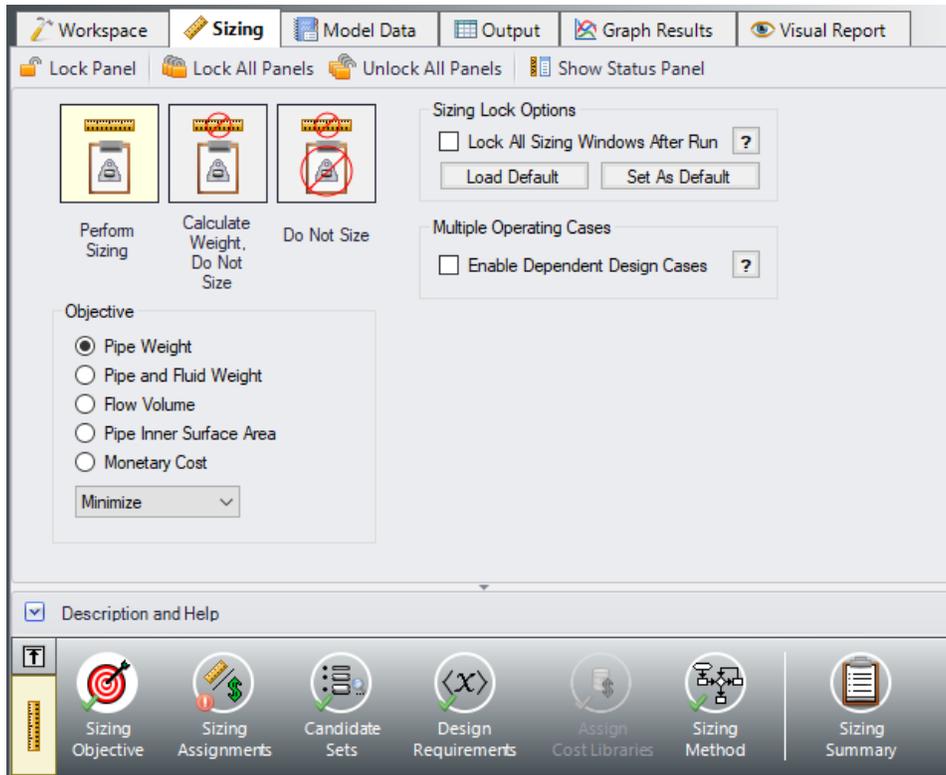


Figure 3: Sizing Window showing common sizing panel features

A. Sizing Objective

The Sizing Objective panel is used to set the Sizing Option, which is the type of calculation that will be performed, and to select an Objective which will be used in the sizing process.

For the Sizing Option, Perform Sizing should generally be selected, since this is the primary function of the ANS module. However, the other two options may be used for troubleshooting or informational purposes. For example, the Calculate Cost, Do Not Size option may be useful to calculate an initial cost for the system, or to verify your cost libraries. The Do Not Size option allows the model to be run normally in AFT Fathom without requiring any cost or sizing information. This is useful to preserve the information already entered in the Sizing window.

➤ **Select Perform Sizing** (if not already selected) to begin configuring the sizing settings.

Note: The Sizing Option can always be seen on the box on the left of the Sizing Navigation panel. Clicking this box will bring you to the Sizing Objective panel, where this status can be changed.

The easiest way to size the system is to set the objective to a non-monetary value such as pipe weight or flow volume rather than directly analyzing monetary cost. Here we will minimize flow volume. The flow volume is merely the sum of the internal volumes of all sized pipes. As the flow volume is minimized, the pipe material, and therefore cost, is minimized along with it. Typically it is recommended to use the pipe

weight option, as this will have the best correlation to cost. However, certain systems, such as ducts may be better sized using flow volume.

Although we will not do so here, the ANS module can also minimize the monetary cost of the system. This is more powerful than sizing for flow volume, but also requires more time investment in setting up cost libraries.

➤ **For the Objective, choose Flow Volume**, and make sure Minimize is selected from the drop-down list. The Sizing Objective panel should now appear as shown in Figure 4.

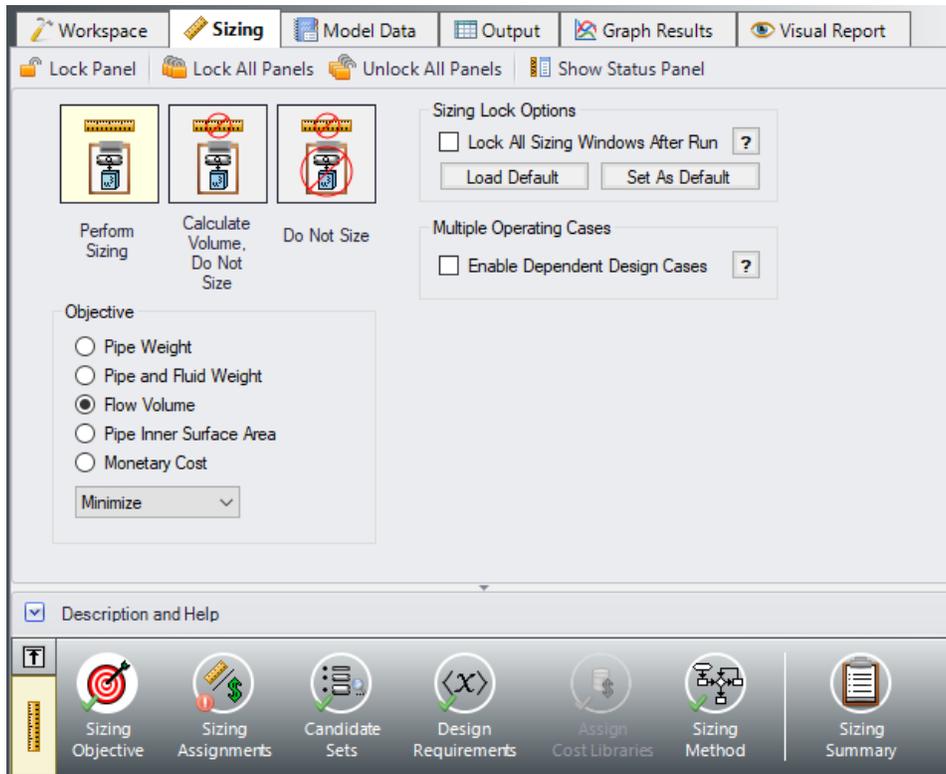


Figure 4: Sizing Objective set to minimize flow volume

B. Sizing Assignments

On the Sizing Navigation panel at the bottom of the window select the Sizing Assignments button. The Sizing Assignments panel allows the user to define what objects will be sized in the model, and what will be included in the cost calculation without being sized. Common Size Groups for pipes, or Maximum Cost Groups for pumps/control valves can be created on this panel as well when appropriate.

Let's consider the Sizing Assignments for the pipes, which should be displayed by default when we open this panel.

In the Sizing and Volume Options table there are several options under the categories to Automatically Size, or Do Not Size the pipe.

If an Automatically Size option is selected, the ANS module will treat the pipe diameter as a variable and vary it according to certain criteria that will be discussed shortly.

For a new system, typically all pipes will be desired to be sized and included in the cost, which is being calculated in terms of volume in this case. If you are instead analyzing the possible replacement of existing pipes, it may be better to size the pipes, but only include the cost if the size changes from that which exists.

If Do Not Size is selected, the pipe will retain the settings currently set in the Workspace. Why would one choose to not size a pipe? There could be a number of reasons, but one good reason is that the pipe represents a pipe in an existing system and the design does not allow the replacement of that pipe with a new one. Therefore its diameter is fixed, and sizing the pipe would serve no purpose.

Another reason may be if a certain size is necessary for the design due to certain requirements, in which case the pipe cost can be included without sizing the pipe by choosing Include in Cost.

This model is a new system, so all pipes will be sized and included in the cost.

➤ **Select Always Include in Volume** for all three pipes (Figure 5).

Since we want to size the pipes independently in this model, we do not need to make any changes to the Pipe Grouping section of the table, so this panel is complete.

Note: For models which have junctions that can be sized, a Junctions button at the top of this panel will be available to set the Junctions Sizing Assignments. In this case we have chosen to analyze flow volume. While this value can easily be calculated for the pipes, Fathom does not have enough information to calculate a flow volume for the branch or the reservoirs, which means that they cannot be sized using this method, and the Junctions tab is unavailable. This same limitation exists when using other non-monetary objectives as well, such as pipe weight. If the size of a junction will have a large impact on the system cost, such as a pump, Monetary Cost should be used as the Objective.

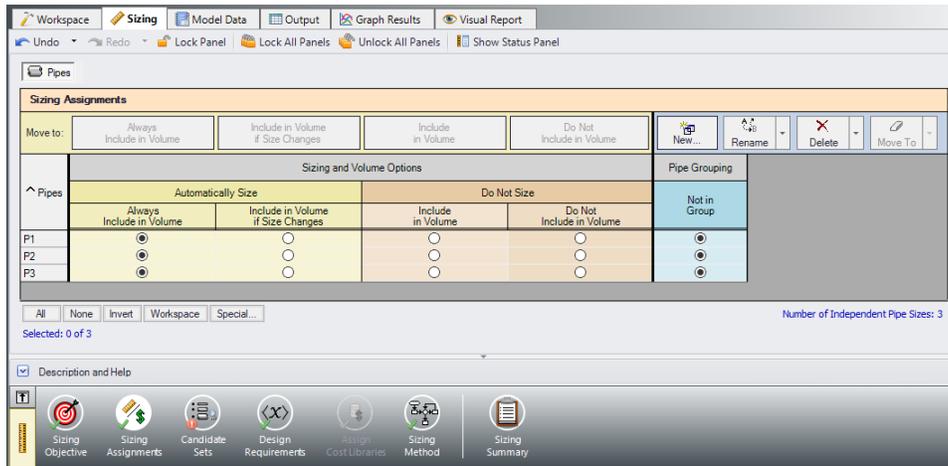


Figure 5: Sizing Assignments panel for Three Reservoir model

C. Candidate Sets

Click on the Candidate Sets button to open the Candidate Sets panel.

Since commercial piping is limited to certain sizes, the ANS module needs a list of possible sizes from which to choose. This list is called a *Candidate Set*.

So as not to limit the sizing unnecessarily, the candidate sets should include both smaller and larger pipes than your anticipated final size. If you make the candidate set too small, you may limit the ability of the ANS module to find the best sizing. It is better to make the candidate set too large than too small. Experience applying the ANS module to actual systems will help you choose appropriate candidate sets. If after obtaining a solution you find that one or more of the sized pipes is at the extreme of the candidate set a warning will appear, and it is recommended to expand the defined candidate set.

To create a Candidate Set, do the following:

1. Under Define Candidate Sets, click **New**.
2. Give the set the name *All STD* and click **OK**.
3. From the drop down list choose **Steel - ANSI** if it is not already selected.
4. At the bottom of the Select Pipe Sizes window, make sure the Sort option selection is **Type, Schedule, Class**.
5. In the Available Material Sizes and Types list on the left, **select (not expand) STD** so that the schedule name is now highlighted.
6. Click **Add** to add all STD pipe sizes to the list on the right.
7. Click **OK**.

The All STD set will now appear.

We will apply the All STD set shortly. First, let's take a moment to understand what we have just done. We have created a candidate set that includes all pipe sizes in the library that are Steel - ANSI STD. At the low end this includes 1/8 inch pipe, and at the high end 48 inch pipe. This candidate set contains pipe sizes from a library of steel ANSI pipe, and not just specific pipes from the model. After we apply this candidate set to a specific pipe in the model and the sizing is run, the ANS module will select the best pipe size from this set of pipe sizes to achieve the objective.

We now need to define which pipes will use this Candidate Set during the sizing calculation. Each pipe that is being sized must have a Candidate Set assigned to it. Under Assign Candidate Sets to Pipes, **set each pipe in the model to use All STD** by selecting the radio button under this candidate set for each of the pipes. The Candidate Sets should now be fully defined and assigned to the appropriate pipes, as can be seen in Figure 6.

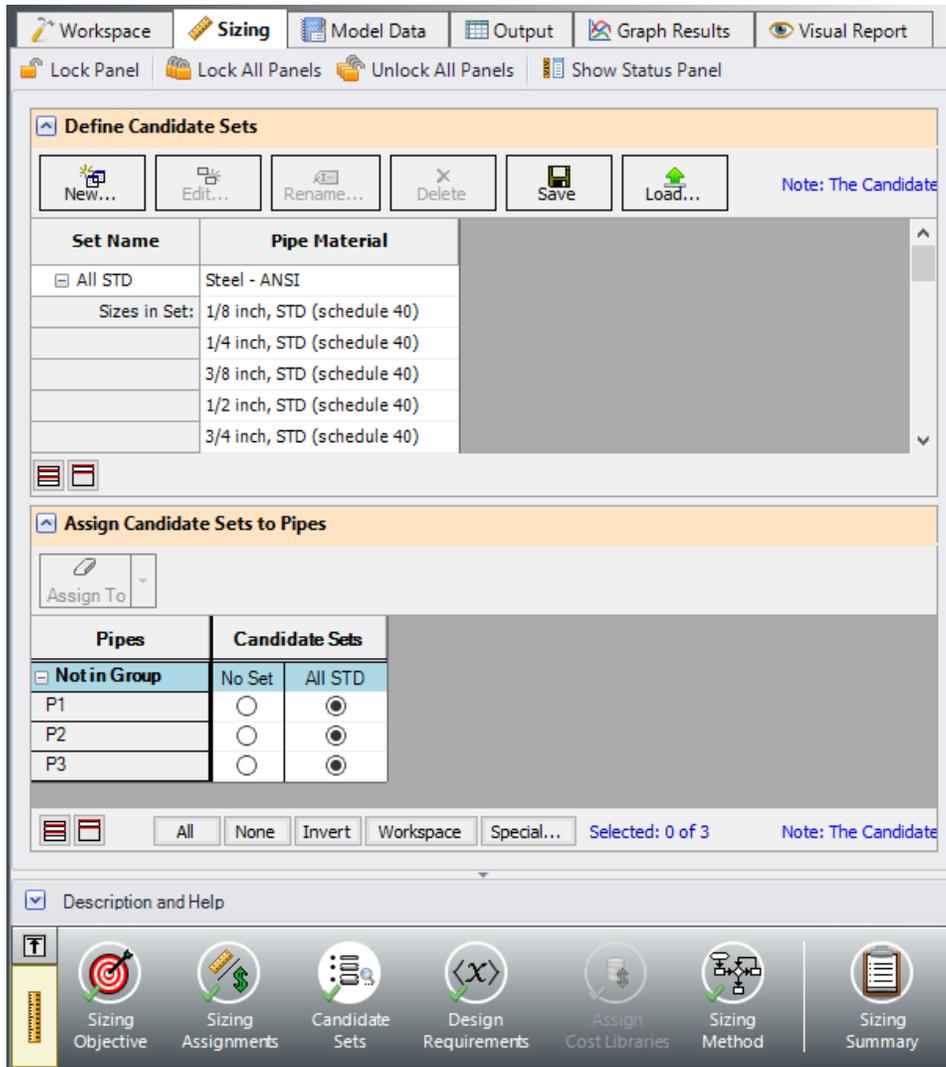


Figure 6: Candidate Sets window fully defined for the Three Reservoir model

D. Design Requirements

Select the Design Requirements button. If we do not define any design requirements for the model, the ANS module will automatically choose the smallest pipe size in the Candidate Set, since it has the smallest flow volume. Along with the smallest pipes we may get unacceptably low flow rates, or unacceptably high velocities and pressure drops. To maintain acceptable system operating conditions, we need to set design requirements.

We are going to define two design requirements for this case. Before adding any requirements, let's look more closely at the results of the model we are starting with. All pipe diameters are nominal 8 inch, and the results show that the water flows from reservoir J1 to reservoirs J2 and J3. The flowrate out of J1 is 542.3 m³/hr, and the flowrates to J2 and J3 are 171.3 and 371.0 m³/hr, respectively. Let's assume that there is a requirement to draw a minimum of 600 m³/hr from J1, and to deliver at least 150 m³/hr each to

J2 and J3. What is the combination of pipe sizes which achieve the desired objective while still meeting these demands? In our case, we are defining the objective as minimizing the internal volume of the pipes (i.e., length times internal cross-sectional area).

We need to create two Design Requirements that represent the flow requirements at the reservoirs in order to answer this question. We will do this by defining Design Requirements for the inlet/outlet flowrates of the pipes where they connect to each of the Reservoirs. To do this:

1. Make sure that the **Pipes** button is selected.
2. Click **New** under Define Design Requirements.
3. Enter a name when prompted: *Min Flow Supply*. Click **OK**.
4. A new row will appear in the Pipe Design Requirements table. In this row, select **Volumetric Flow Rate** as the Parameter.
5. Choose **Minimum** for Max/Min.
6. Enter 600 m3/hr.
7. Now repeat the above process to define *Min Flow Delivery* with a **Volumetric Flow Rate Minimum** of **150 m3/hr**.

To be functional, we need to apply these Design Requirements to the correct pipes. There are no limits to the number of Design Requirements that can be defined and applied.

In the Assign Design Requirements to Pipes section, expand the pipe list. Select the check box for Min Flow Supply next to pipe P1, and Min Flow Delivery next to pipes P2 and P3. This will set the minimum flow rate coming out of Reservoir J1 and going into Reservoirs J2 and J3 as described above. The fully defined and assigned Design Requirements are shown in Figure 7.

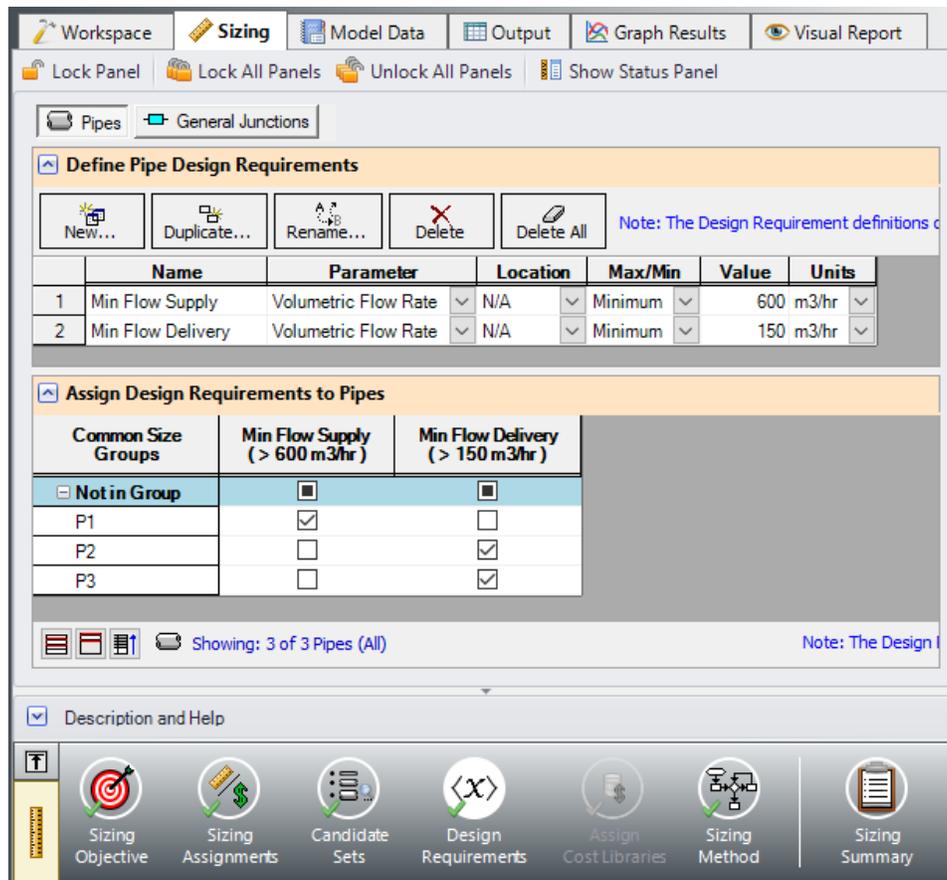


Figure 7: Design Requirements window with the minimum flow requirements defined

E. Assign Cost Libraries

When the Sizing Objective has been defined as Monetary Cost, it is necessary to create and assign cost libraries for the automated sizing, which can be done in the Assign Cost Libraries panel. Since we have defined the objective as Pipe Flow Volume, we will not need to assign any cost libraries, and this button is grayed out.

F. Sizing Method

Select the Sizing Method button to go to the Sizing Method panel.

The Sizing Method panel is used to set up the calculation methods for sizing the system. You can select whether discrete or continuous sizing will be used, and which method will be applied.

If Continuous Sizing is selected, the ANS module will ignore the defined Candidate Sets and report the ideal hydraulic diameter for the pipes being sized, which will likely not match any of the possible chosen commercial sizes. While this is not useful as a final solution, this may be helpful as a baseline to check the final solution using the provided discrete methods. Discrete Sizing will typically perform a continuous

sizing calculation as a basis, after which multiple discrete sizes above and below this solution will be evaluated to find the ideal sizing based on the provided Candidate Set.

➤ **For this model, use Discrete Sizing.**

For the Search Method, the ideal method will often change based on the number of independent pipe sizes (shown in the Sizing Assignments panel on the bottom right), number of Design Requirements, and feasibility of the initial system design. The Help file provides more information on the strengths and weaknesses of each method. It is generally recommended to run the sizing with more than one method, as it is often not obvious which method will be most effective for each system. The suggest Method button can be used as a guide for which method to start with.

For a simple model such as this one, the MMFD or SQP method should be appropriate since there are only a few Design Requirements, and only three pipes are being varied.

➤ **Choose the default Modified Method of Feasible Directions (MMFD).**

Step 5. Run the Model

Select Run from the Analysis menu. While the model is running, the Solution Progress window shows the Sizing Calls to Solver. This is how many times a complete hydraulic analysis was run.

The solver also displays the Current Cost and Best Feasible Cost, which will display the last calculated value for the cost, as well as the Best Feasible Cost which has been found so far (Figure 8). The solver will continue to iterate using the defined method until it finds the ideal sizing. For the selected method, the solver will first perform a continuous sizing to find a starting point, then test discrete solutions close to this continuous solution to find the final, discrete solution.

Note: At any time during the run, the user can pause the solution and use the Other Actions button to accept the currently displayed Best Feasible Cost as the final solution. For this example the solution is found and displayed quickly, but for larger systems with many pipes being sized, it may be more productive to pause the solution and accept the current solution in order to save time, especially if the displayed cost does not appear to be decreasing much further.

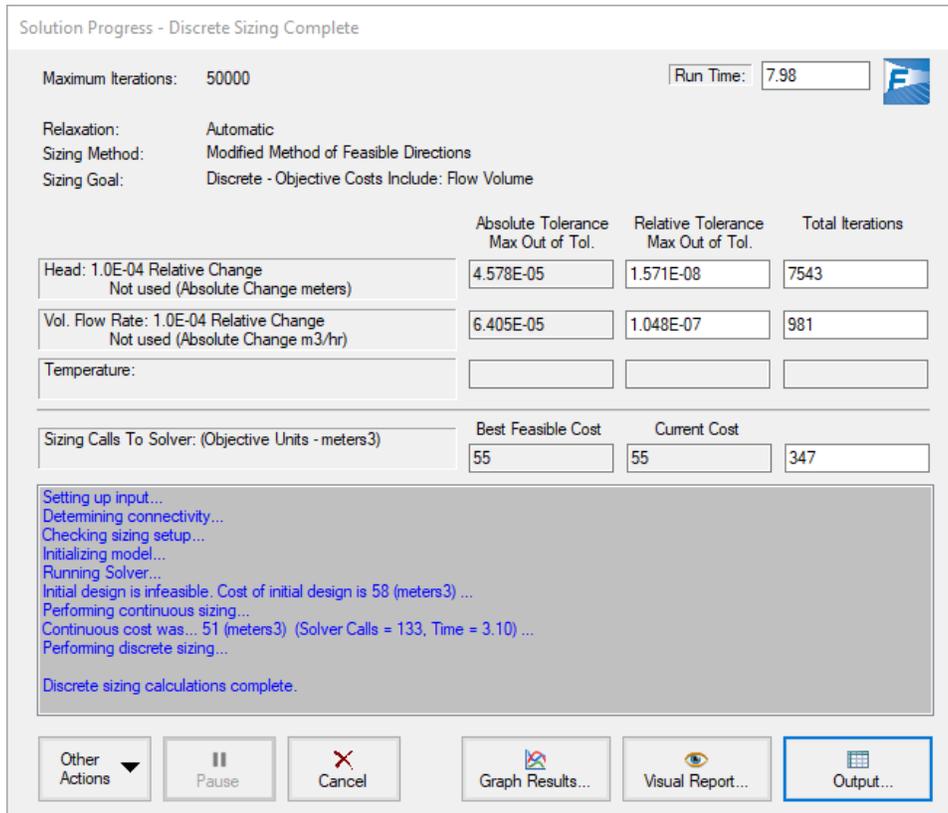


Figure 8: Solution Progress Window showing the progress of the sizing calculation

Step 6. Review the Sized Results

Click the View Output button to see the results. The General section shows the Cost Report, which indicates the volumes for the final solution. As shown in Figure 9, the minimum total volume identified by the ANS module was 55 meters³. In the Pipes section of the output on the Sizing tab, the final pipe sizes of 10, 6, and 8 inches are shown (Figure 9).

Now switch to the Pipe Design Requirements tab in the Pipe section (Figure 10). We can confirm that the calculated design is feasible, since all of the Design Requirements were met. That is, the flow from J1 (as shown in pipe P1) exceeded 600 m³/hr, and the flows to J2 and J3 exceeded 150 m³/hr. Notice that some of the Design Requirements are highlighted, indicating that they were active. An active design requirement is defined as a requirement that had an impact on the solution. If an active design requirement were disabled, a different solution would be found. Yellow highlights indicate that the design requirement was active in the final solution, while blue highlights indicate that the requirement was active when the continuous sizing was performed, but became inactive when the solver switched to discrete sizing.

You can also display the adjusted pipe sizes and resulting flow rates in the Visual Report.

Table Units: meters ³					
Type	Name	Flow Volume			
Total of All Shown Costs		55			
Items In Sizing		55			
Items Not In Sizing		0			
Pipe Subtotal		55			
o	P1	Pipe	Pipe	15	
o	P2	Pipe	Pipe	11	
o	P3	Pipe	Pipe	29	

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (cm)
1	Pipe	Steel - ANSI	10 inch	STD (schedule 40)	25.45
2	Pipe	Steel - ANSI	6 inch	STD (schedule 40)	15.41
3	Pipe	Steel - ANSI	8 inch	STD (schedule 40)	20.27

Figure 9: Output Window with Cost Report and Pipe Sizes for Three Reservoir Model

Pipe	Volumetric Flow Rate Minimum > 600 m ³ /hr [Min Flow Supply]	Volumetric Flow Rate Minimum > 150 m ³ /hr [Min Flow Delivery]
1	611.2	
2		166.5
3		444.7

Figure 10: Design Requirements results for Three Reservoir Model

Transferring Sized Results

When performing sizing, the Transfer Results to Initial Guesses feature on the Output window Edit menu takes on new meaning. When used in a non-sizing context, this feature takes the solved hydraulic results (i.e., pressures, flows rates, etc.) and assigns them to the initial guess values for the pipes and junctions.

When you select Transfer Results to Initial Guesses after automated sizing, you will see a dialog window appear like that shown in Figure 11. The ANS module allows you to transfer the hydraulic results, the

sized pipe diameters, or both. If you transfer the sized pipe diameters, each pipe's input diameter (back on the Workspace) will become the sized diameter.

This feature allows the sizing results to be saved to the model for use in further analysis as needed.

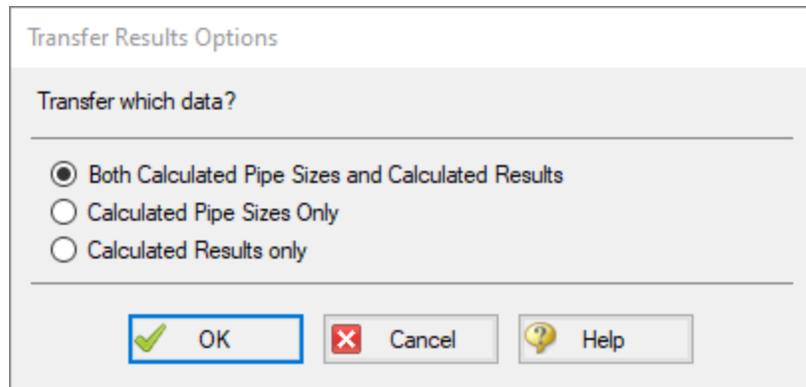


Figure 11: After sizing, the *Transfer Results to Initial Guesses* feature transfers both the hydraulic results and sized pipe size results

Conclusion

You have now used performed a network sizing to minimize flow volume with the ANS module.

Control Valve - ANS

Summary

This example will show how to use the ANS module to size a system based on pipe weight. For many systems the necessary cost data is not available to directly minimize cost, so minimizing system parameters such as weight or flow volume is useful. By minimizing parameters such as these we minimize pipe diameter, which directly correlates to minimizing pipe costs.

We will create a system that feeds water from an elevated reservoir to a lower reservoir at a specified flow rate. A control valve will be used to control the flow rate and can be placed anywhere within the line. We will need to determine what size of piping to use in order to achieve a minimum pressure drop of 82 kPa across the control valve.

Note: This example can only be run if you have a license for the ANS module.

Topics Covered

- Sizing using pipe weight
- Creating a Common Size Group
- Defining Control Valve Design Requirements

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the [Beginner - Three-Reservoir - ANS](#) example, and is familiar with the basics of ANS analysis.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Control Valve - ANS - Initial.fth***
- ***Metric - Control Valve - ANS - Final.fth***

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **Metric - Control Valve - ANS - Initial.fth** example file. The workspace should look like Figure 1 below:

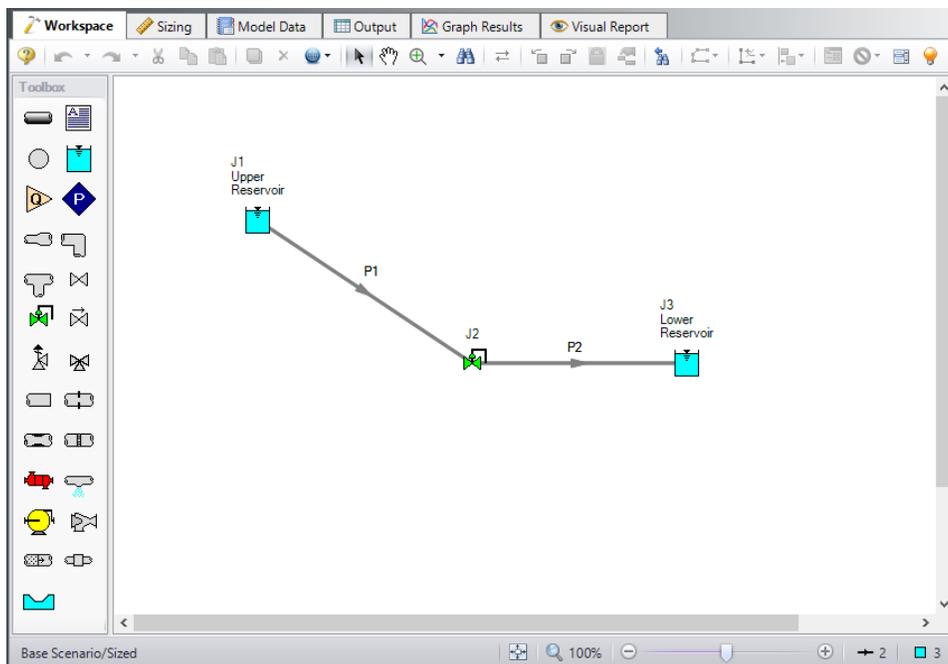


Figure 1: Control Valve System Layout

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate ANS and select Network to enable the ANS module for use. A new group will appear in Analysis Setup titled Automatic Sizing. Click OK to save the changes and exit Analysis Setup. A new Primary Window tab will appear between Workspace and Model Data titled Sizing. Open the Analysis menu to see the new option called Automatic Sizing. From here you can quickly toggle between Not Used mode (normal AFT Fathom) and Network (ANS mode).

Step 4. Configure Sizing Settings

In this model we are going to perform sizing to minimize the pipe weight (therefore minimizing cost), while maintaining a minimum pressure drop of 82 kPa across the control valve.

To do this, first go to the Sizing window by clicking on the Sizing tab, or from the Windows menu.

A. Sizing Objective

The Sizing Objective panel should be selected by default from the Sizing Navigation Panel along the bottom of the window. Input the following:

1. Sizing Option = Perform Sizing
2. Objective = Pipe Weight
3. Option = Minimize

B. Sizing Assignments

On the Sizing Navigation Panel at the bottom of the window select the Sizing Assignments button.

In this case we are sizing a new pipeline, so we want to size all pipes in the model.

➤ **Under Automatically Size select Always Include in Weight** next to both pipes.

For this system it is required that both pipes be the same size. We will need to add both pipes to a Common Size Group to specify this. Do the following:

1. Click **New** above Pipe Grouping
2. Enter the a name *Main Pipes* and click **OK**. A new section will now appear with the Main Pipes group listed under Common Size Groups.
3. **Add both pipes** to this group by clicking the corresponding radio buttons (Figure 2).

Note that as many Common Size Groups as necessary can be created. Selection tools are available at the bottom of the window to help assign multiple pipes to a Common Size Group for larger models. Additionally, pipes can only be assigned to a group once they are set to be included in sizing.

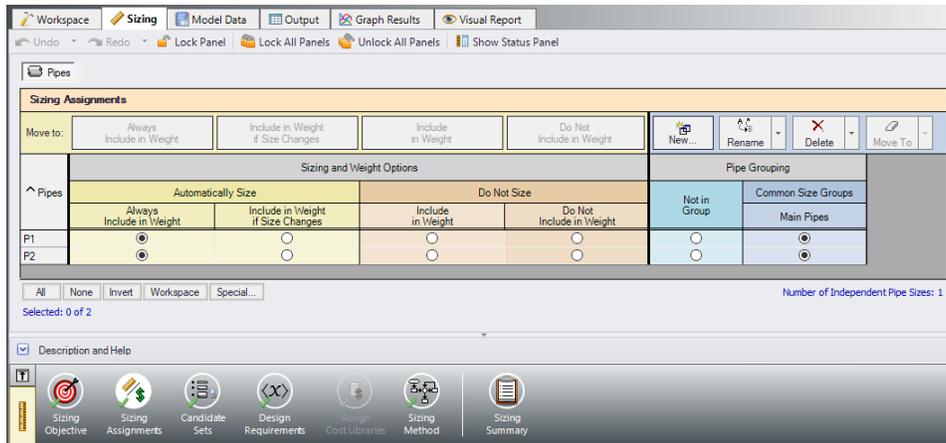


Figure 2: Pipes P1 and P2 settings in the Sizing Assignments window

C. Candidate Sets

Click on the Candidate Sets button to open the Candidate Sets panel.

1. Under Define Candidate Sets, click **New**.
2. Give the set the name **STD Pipes** and click **OK**.
3. In the Select Pipe Sizes window, choose **Steel - ANSI** from the drop-down menu if not already selected.
4. Make sure the Sort selection is for **Type, Schedule, Class** at the bottom of the window.
5. In the Available Material Sizes and Types on the left, **expand** the STD pipe sizes list.
6. Double-click each of the sizes **from 1 inch to 12 inch** to add them to the list on the right.
7. Now click **OK**.

The STD Pipes set will now appear as a Candidate Set at the top.

We now need to define which pipes will use this Candidate Set during the sizing calculation.

➤ Under **Assign Candidate Sets to Pipes**, set the **Main Pipes Common Size Group** to use the STD Pipes Candidate Set. The Candidate Sets are now fully defined and assigned to the appropriate pipes, as can be seen in Figure 3.

Figure 3: Candidate Sets panel fully defined for the Control Valve model

D. Design Requirements

Select the Design Requirements button.

For this model we only have one Design Requirement, which is for the minimum pressure drop across the Control Valve. To define this requirement:

1. Select the **Control Valves** button at the top of the window.
2. Click **New** under Define Control Valve Design Requirements.
3. When prompted, enter the name *Min Pressure Drop*.
4. Next to Min Pressure Drop in the table, select **Pressure Drop Static** as the Parameter.
5. Choose **Minimum** as the Max/Min option.
6. Enter 82 kPa.

Now we need to apply the defined Design Requirement to the Junction.

➤ **Check the box next to the Control Valve J2** in the Assign Design Requirements to Control Valves section.

Note: Control valves must always have a design requirement specified for the minimum pressure drop/head, or the maximum Cv/percent open at the valve to ensure that reasonable operating conditions exist at the control valve. If no Design Requirements are applied, the ANS module will allow the control valve to control to any pressure required to meet the control setpoint, even if this requires the control valve to add pressure to the system. To avoid this in the final solution, a Design Requirement should be applied.

E. Assign Cost Libraries

When the Sizing Objective has been defined as Monetary Cost, it is necessary to create and assign cost libraries for the automated sizing, which can be done in the Assign Cost Libraries panel. Since we have defined the objective as Pipe Weight, we will not need to assign any cost libraries, and this button is grayed out.

F. Sizing Method

Select the Sizing Method button to go to the corresponding panel.

Make sure **Discrete Sizing** is selected, since we are sizing based on commercial pipe sizes.

For the Search Method in a simple model such as this one, the MMFD or SQP method should be appropriate since there is only one Design Requirement, and only one pipe size is being varied. Choose the default **Modified Method of Feasible Directions (MMFD)**.

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Review the Sized Results

The output is shown below in Figure 4. We have shown all three sections of the output window with the General, Pipes, and Junctions sections. In the General Section, the Cost Report tab shows the minimized pipe weight for each of the pipes that were sized based on the design requirements.

In the pipe section, the Sizing tab is selected to show the final pipe sizes that were chosen. The ideal pipe size was found to be 6 inch pipe for this case. In the Junction section, the CV Design Requirements tab is shown. It can be seen that the Control Valve Design Requirement was met. The Design Requirement is colored blue, indicating that it was active in the final solution.

The screenshot shows three sections of the software output window:

General - Cost Report

Table Units: kg		Type	Name	Pipe Weight
Total of All Shown Costs				1,694
Items In Sizing				1,694
Items Not In Sizing				0
Pipe Subtotal				1,694
o	P1	Pipe	Pipe	847
o	P2	Pipe	Pipe	847

Pipes - Sizing

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (cm)
1	Pipe	Steel - ANSI	6 inch	STD (schedule 40)	15.41
2	Pipe	Steel - ANSI	6 inch	STD (schedule 40)	15.41

Junctions - CV Design Requirements

Jct	Pressure Drop Static Minimum > 82 kPa [Min Pressure Drop]
2	85.53

Figure 4: Output window for Control Valve Sizing

Step 7. Change the Control Valve Location

Now we would like to try something to further improve the system. For the first cut, we assumed that the control valve was in the middle of the pipe and that the pipes had to be the same size. If the pipes do not

have to be the same length or size, we can further minimize the pipe weight and theoretically the cost of the piping.

To do this we will create another scenario.

1. In the Scenario Manager, **create a child** from the Base Scenario and name it *Unequal Pipes*. There should now be a check mark icon next to the scenario name to show that it is active.
2. On the Workspace open the Pipe Properties window for pipe P1 and change the length to **45 meters**.
3. Open the Pipe Properties window for pipe P2 and change the length to **15 meters**.
4. Go to the **Sizing** tab and click the **Sizing Assignments** button in the Sizing Navigation panel.
5. In the Pipe Grouping section of the table set **both** pipes to **Not in Group**. This will allow their sizes to be varied separately.

The scenario is now complete.

Step 8. Run New Scenario and Review Results

Now, we will run this scenario and see if we have minimized the pipe weight further. Select Run from the Analysis menu and allow the ANS module to size the system. When the analysis is complete, select Output to view the results.

The output window is shown below in Figure 5. We can see that pipe 1 was now chosen as 5 inch but pipe 2 was still chosen as 6 inch. Since pipe 1 was smaller the pressure drop across the pipe was greater, so the pressure drop across the valve was reduced from 85.53 to 82.22 kPa. The pipe weight was reduced from 1,694 to 1,402 kg.

Table Units: kg					
	Type	Name	Pipe Weight		
Total of All Shown Costs			1,402		
Items In Sizing			1,402		
Items Not In Sizing			0		
Pipe Subtotal			1,402		
o	P1	Pipe	Pipe	979	
o	P2	Pipe	Pipe	424	

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (cm)
1	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	12.82
2	Pipe	Steel - ANSI	6 inch	STD (schedule 40)	15.41

Jct	Pressure Drop Static Minimum > 82 kPa [Min Pressure Drop]
2	82.22

Figure 5: Output window for Unequal Pipes Scenario

Conclusion

This example has demonstrated how simple it is to perform an automated sizing to design a system for a set of requirements. Pipe weight may not be directly proportional to pipe cost but it can be implemented very quickly when cost data is not available.

Cooling System - Creating Libraries - ANS

Summary

This example focuses on creating Engineering and Cost Libraries for a system, then connecting the libraries to the model in order to obtain accurate results based on the cost data.

Note: This example can only be run if you have a license for the ANS module.

Topics Covered

- Using external libraries
- Building engineering and cost libraries
- Using Scale Tables for cost
- Connecting and using existing engineering and cost libraries

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

This example is an addendum to the [Cooling System - ANS](#) example, but can be worked separately from the example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Cooling System.fth***
- ***Metric - Cooling System Libraries - ANS.fth***
- ***Creating Libraries - Cost Templates.xlsx*** - template file for building cost libraries
- ***Creating Libraries.dat*** - completed engineering library
- ***Creating Libraries Costs.cst*** - completed cost library for ***Creating Libraries.dat***
- ***Steel - ANSI Pipe Costs.cst*** - completed cost library for **Steel - ANSI** pipes

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **Metric - Cooling System.fth** example file and save it as a new file. We will not need the existing child scenarios for this example: right click on the Base Scenario in the Scenario Manager and select Delete All Children.

For the analysis we will be creating the library files for a completed model file to focus solely on using libraries as part of the costing and/or sizing process.

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate ANS and select Network to enable the ANS module for use. A new group will appear in Analysis Setup titled Automatic Sizing. Click OK to save the changes and exit Analysis Setup. A new Primary Window tab will appear between Workspace and Model Data titled Sizing. Open the Analysis menu to see the new option called Automatic Sizing. From here you can quickly toggle between Not Used mode (normal AFT Fathom) and Network (ANS mode).

Step 4. Creating an Engineering Library

If it is desired to size any of the junctions in the system, such as the pumps or elbows, the first step is to create an engineering library containing information for each of the junctions that will be sized. This engineering library can later be used to attach a cost library for the junctions. Engineering libraries can also be connected to future models to reuse any of the junctions in future systems.

Note: This step is not required if it is only desired to include pipe costs for the pipe material information included in Fathom, since the cost libraries for the pipes can be built off of the default Fathom Pipe Material Libraries.

To add a junction to an engineering library:

1. Define the junction in the workspace.
2. **Select** the junction, go to the **Library** menu, and choose **Add Junction to Library**.
3. Give the junction a meaningful name and add it to the desired library by selecting it from the list.

For the purpose of this example the model has already been fully defined, so we will just need to add the junctions to the library which we will be calculating the cost for. Follow the steps above to add Pump J4 and Bend J3 to the LOCAL USER LIBRARY. Give the pump the name *Sizing 680 m3/hr* and the bend the

name *STD Bend*. In this case only one of each junction needs to be added to the library, since each of the pumps and bends in the model are identical. If we had multiple pump models, bend types, etc. then we might need to add multiple pumps, bends, etc.

The local user library is the default location where user junctions are placed for the local machine. If the junctions are stored in the local user library, it is possible to run all of the calculations, and attach a cost library for the junctions. However, this restricts the use of the saved data, as the local user library cannot be shared with other users or be moved to different machines. This is why it is often preferred to place the junctions into an external library. Follow the steps below to move the junctions:

1. Go to the **Library** menu and open the **Library Manager**.
2. At the bottom, select **Create New Library**, and give it the filename *Creating Libraries*. It is important to use a descriptive filename to avoid confusion later if the library needs to be reconnected, or sent to someone else.
3. Enter a Library Description. This is the name that will be displayed within the Library Manager itself. The description may be different from the file name, but it is typically recommended to make these the same to avoid confusion later. Use the name *Creating Libraries* for the description.
4. In the LOCAL USER LIBRARY, expand the Junctions option. The window should appear similar to Figure 1.
5. For both junctions that we added earlier, right-click the name, then click **Move Content**. In the Select Library window, click *Creating Libraries* and then click **Select**. This moves the junction from the local user library to the new external library. The pump and bend should now be visible within their respective sections on the right side of the window.
6. Select **Close** to complete the process. If you open the folder where the library was created in Step 4, the library file should now be available.

We are now ready to enter the cost data for the junctions in the model.

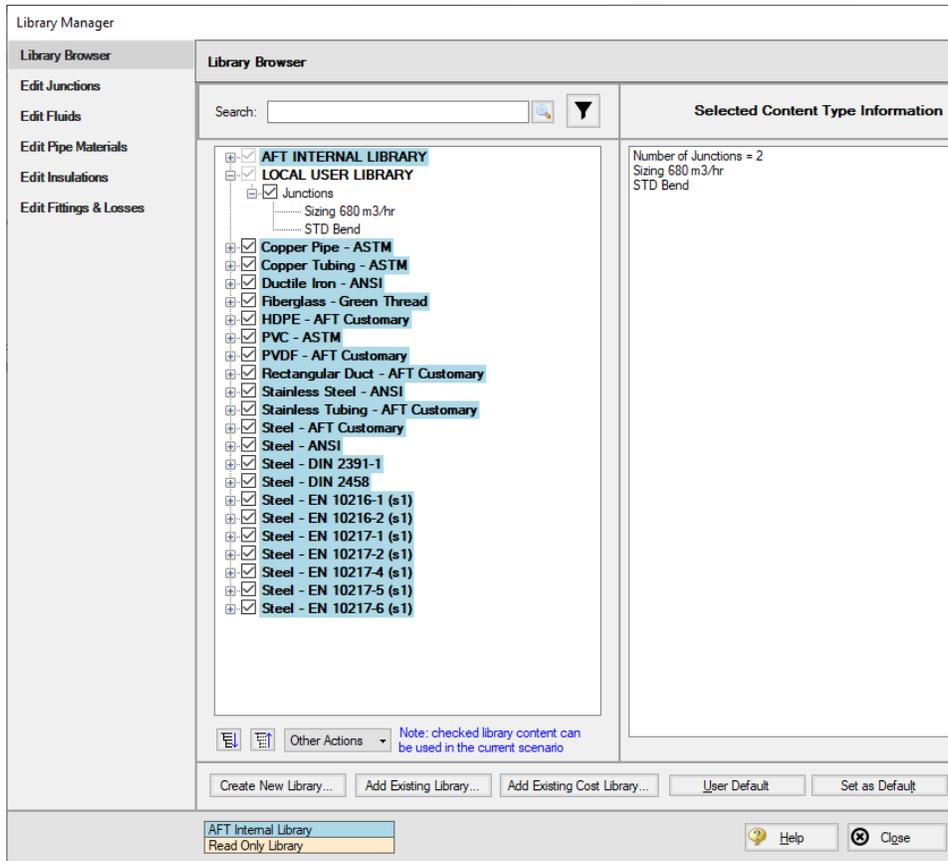


Figure 1: Edit Library tab in the Library Manager with a new external library created for the junctions to be added to

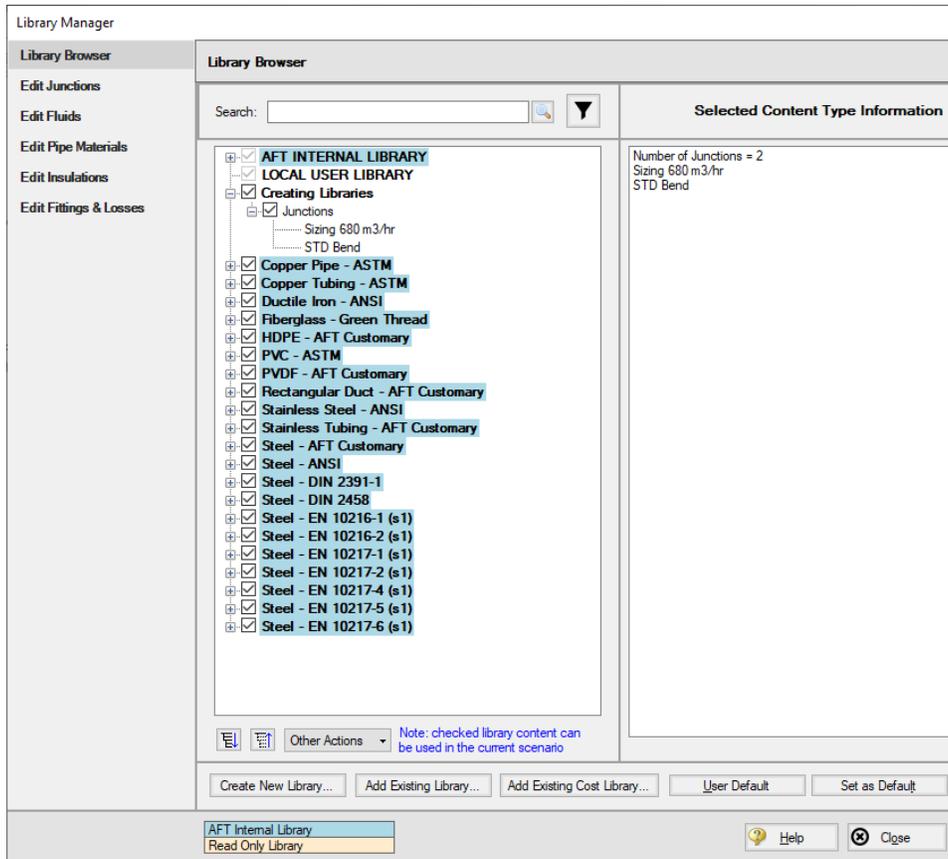


Figure 2: Library Manager showing the junctions moved to the new external library from the local user library

Step 5. Creating a Cost Library for the Equipment

Go to the **Library** menu and select **Cost Library** to open the Edit Cost Library window, which is used to create and view cost libraries. This window can also be accessed from the Sizing window on the Assign Cost Libraries panel. In this case we need to enter new cost data that we have obtained, so click **New**.

The list of Available Libraries should now be displayed as shown in Figure 3. Every cost library must be built off of an existing engineering or pipe material library. Browse to the newly created library, **Creating Libraries**. You may need to use the Browse button to find it if the library is not shown in the list. After you click the library to highlight it, click **Select**.

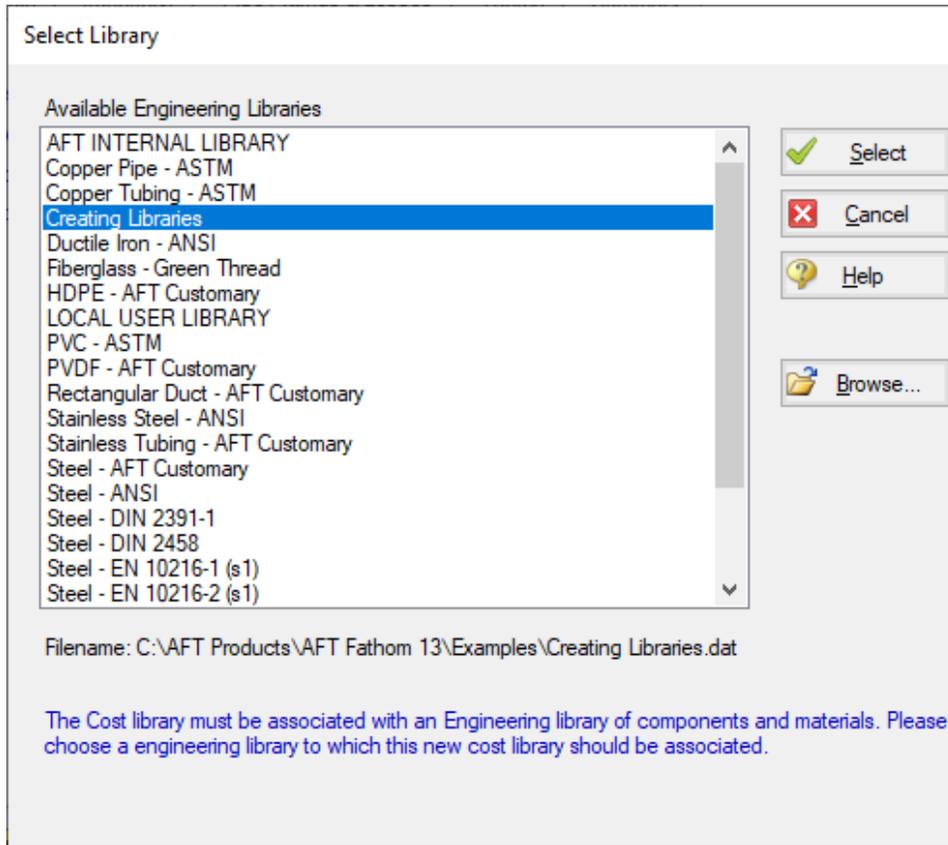


Figure 3: The Available Engineering Libraries list which can be used to build a cost library

From here we will be brought to the General tab. The description field is used as the name displayed within the Library Manager, similar to with the engineering library.

The Notes field can be used to provide information on references, or other details which may be useful for tracking purposes. Notes can be seen when reviewing library contents later using Browse Library Contents.

The other item of note is the Multipliers table. This can be useful to apply multipliers to account for inflation or uncertainty, without needing to calculate the costs directly. Note that multipliers can also be applied within the Assign Cost Libraries panel, so be cautious to avoid duplicate multipliers.

➤ **In the Description field enter Creating Libraries Costs**, as seen in Figure 4. Now let's move on to entering cost data. Click the Junctions tab to view the Junctions overview.

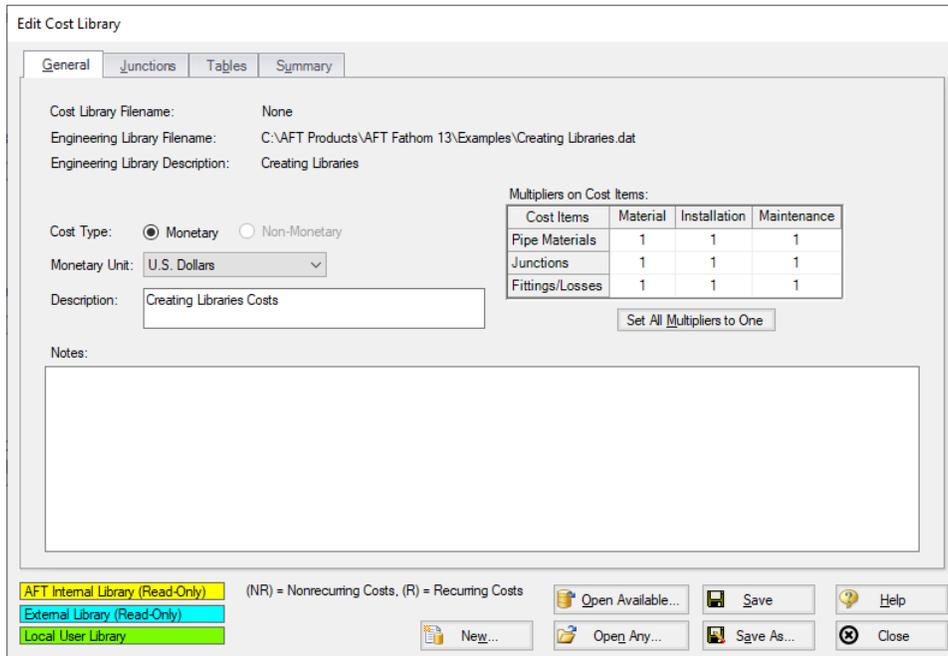


Figure 4: The General tab within the Edit Cost Library window

Within the Junctions tab each individual library item can be viewed under each Junction type. First expand the Pump section by clicking the +, and select the **Sizing 680 m3/hr** junction that was added earlier.

➤ **Click New Cost** to add an item to the table. By default, a cost item will be added that is configured for cost entry per item. Costs can be categorized as Material, Installation or Maintenance Cost Type. This categorization becomes useful if it is desired to exclude certain costs from the calculation or the sizing, but otherwise does not impact how the costs are calculated.

Separately open up the Excel spreadsheet titled **Creating Libraries - Cost Templates.xlsx** found in the Examples folder, and go to the **Junction Costs (Metric)** tab. The cost data for the pumps and bends can be seen here, categorized by cost type.

In this case we are sizing the pump, so rather than having a cost for a specific pump model, we have a range of costs specified based on the power requirement for the pump. We will need to enter this data into the Tables tab first before we can define the cost item on the Junctions tab. Back in AFT Fathom in the Edit Cost Library window, click on the Tables tab at the top of the window.

➤ **Click New Table** to begin defining the data table, and name it *Metric - Pump Material*. Multiple Table Types are offered as the dependent variables for the table, which can either be plotted against a set of multipliers which will be applied to a specific data point, or directly against cost, as can be seen in Figure 5.

The data we have available is the cost for pumps over a range of different power requirements, so choose **Power** and **Cost** as shown below, then click **OK**.

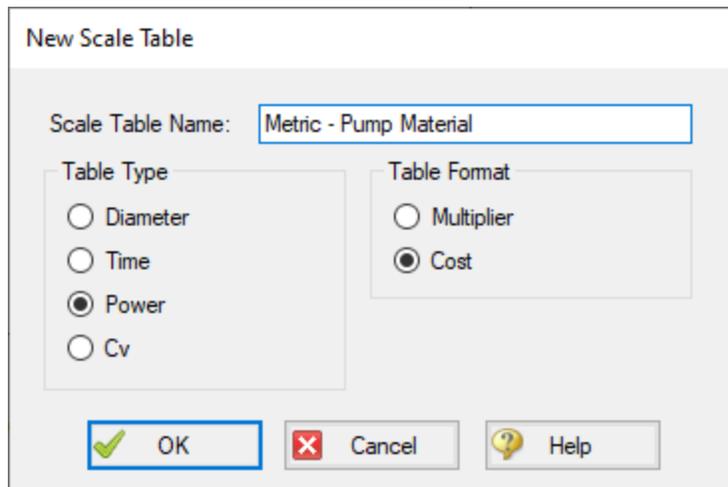


Figure 5: Table type and formatting options when creating a cost library

The new table name will now appear in the Tables list, and an entry section for Power vs Cost will become available. **Copy** the data in the Excel spreadsheet for Pump Material, then click the **Edit Table** button and choose **Paste**. Make sure that the table Units have been set to **kW**. Repeat this process of creating a new table for the pump installation costs using the data from the spreadsheet.

Now that the data tables have been created, we must assign them to the pump junction in the Junctions tab. Click on the **Junctions** tab and make sure that the **Sizing 680 m3/hr** pump model has been selected, then enter the cost information as follows:

1. Description = Purchase
 - This row is optional, but can be useful to provide clarification on the source of the entered cost.
2. Cost Type = Material (NR)
 - Note that NR denotes a non-recurring cost type, while R denotes a recurring cost type.
3. Use Size Table = Table of Costs
4. Multiplier = 1
5. Size Scaling Table = Metric - Pump Material

Click **New Cost** to create a second cost item for the pump and enter the cost information as follows

1. Description = Installation
2. Cost Type = Installation (NR)
3. Use Size Table = Table of Costs
4. Multiplier = 1
5. Size Scaling Table = Metric - Pump Installation

We have specified the cost types for each of the pump data tables, and have assigned them to our Sizing 680 m3/hr model contained in the engineering library. However, we still need to input the data for the Bends.

On the Tables tab, follow the same process as outlined above to **create two more tables** containing the cost information for the bends contained in the spreadsheet. Both tables will use cost for the Table Format, with Diameter as the Table Type. Be careful when filling in the table data to be sure that the correct units are applied for each table. When you are finished there should be four tables total, as shown below in Figure 7.

To assign the tables, return to the **Junctions** tab. Select the **Bend** and **add two cost items**, one applying the material costs table, and one for the installation costs table. When you are done the Bend should have two costs, as shown in Figure 8.

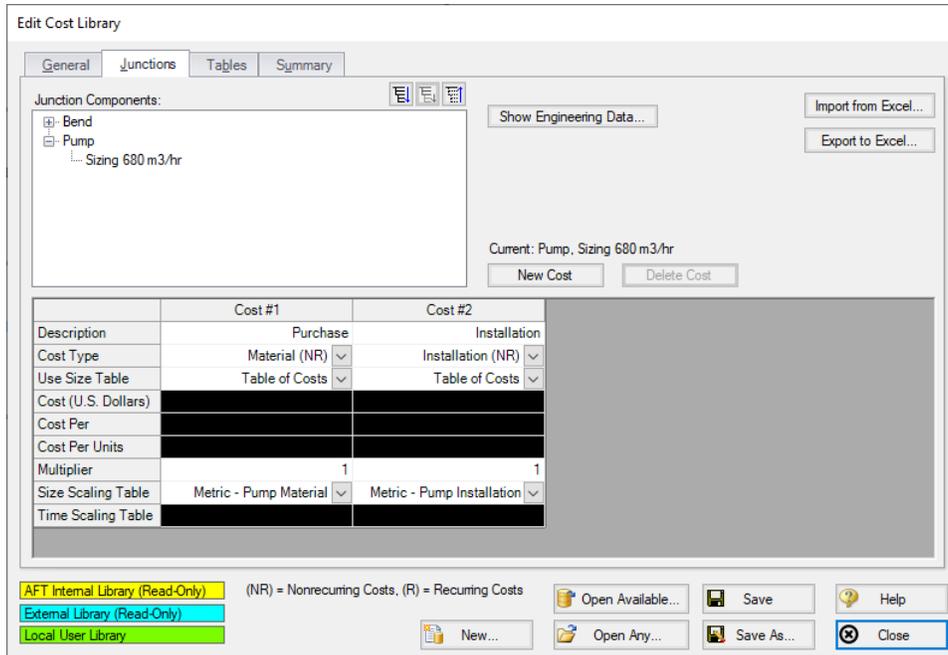


Figure 6: Pump Material and Installation costs assigned using cost tables

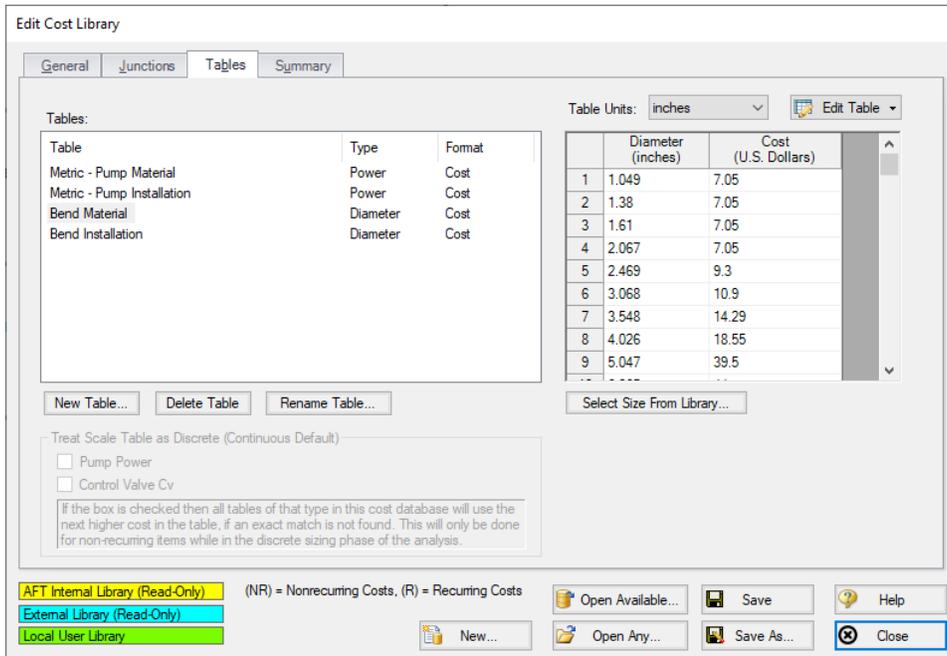


Figure 7: Complete tables list with material and installation cost tables for each of the junctions being analyzed

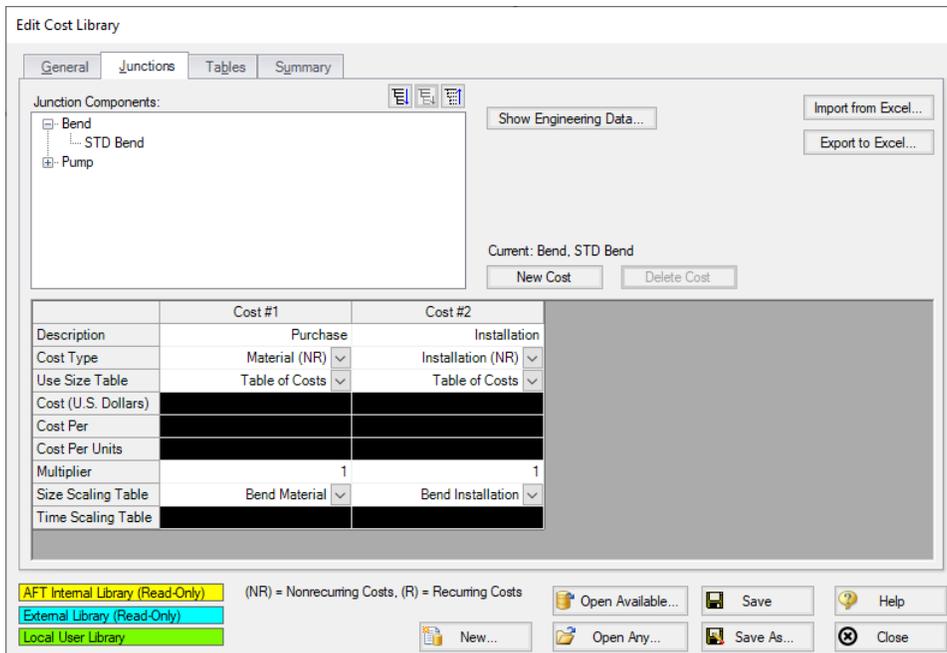


Figure 8: Completed cost items for the bend material and installation costs

Click **Save**, and give the cost library a name. We will use **Creating Libraries Costs.cst** in this case. It is often useful to give the cost library a similar name to the engineering library to make transferring and reconnecting the library faster. Once you have finished saving the library, click Close.

Step 6. Create Cost Library for Pipes

We now have costs for the equipment, but we still need costs for the pipes. In the Sizing tab in the model, go to the Candidate Sets panel by clicking the button. By expanding the existing Candidate set, it can be seen (as stated in the name) that we will need cost information for all STD Steel – ANSI pipe sizes from 1 inch to 36 inches.

1. Return to the **Edit Cost Library** window by going to the Library menu and selecting Cost Library, then click **New**.
2. From the Available Engineering Libraries list, click **Steel - ANSI**, which is the default AFT Steel-ANSI pipe materials library.
3. Click **Select** to choose this library. You should now be on the General tab of the Edit Cost Library window.
4. In Description, give the new pipe cost library the name *Steel - ANSI Pipe Costs*.
5. Go to the **Pipe Materials** tab. In the list section, we will be able to view all of the sizes and types for the Steel - ANSI material. Under Sort choose **Material, Type or Schedule, Nominal Size** so that they will be sorted by type rather than size.
6. Expand the **Steel - ANSI** list, then expand the **STD** list (Figure 9).

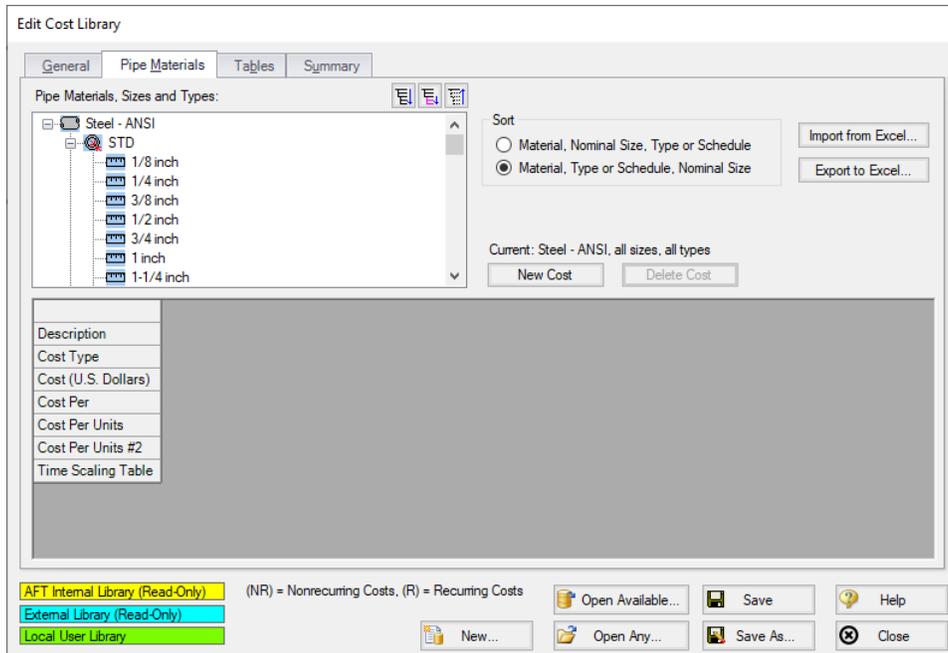


Figure 9: Pipe Materials tab in the Cost Library window showing the available STD sizes

If a size from the list is selected, the hydraulic diameter is displayed, along with a summary of the material, size, and type. The New Cost button will also become active (Figure 9). There are a few methods available when adding the pipe costs. We could individually add the Material and Installation costs for each pipe size by selecting each size from the pipe materials list, then using the New Cost button to add these costs to the table individually. Alternatively, the pipe costs can be entered into a template in Excel, then imported to the Cost Library all at once.

In this case, we have the Pipe costs available in a spreadsheet, so we will use the Excel import/export option. To do this we will first export a template for the desired pipe sizes, then we will enter the cost information in the template and import the updated file as detailed below.

1. Make sure that the STD pipe material type is highlighted in the list, then click **Export to Excel** on the right side of the window. The Excel Export window should be shown as in Figure 10, with all Steel-ANSI STD pipe sizes displayed.
2. Click **Export**, name the sheet, and save it to an easy to find location. The name does not matter, but we will need to be able to find this spreadsheet later to import back into the cost library.
3. When Fathom has finished the export a message will be displayed notifying the user of a successful transfer. Click **OK**, then browse to the exported worksheet and open it. The exported worksheet should contain a list of the selected pipe sizes, with empty columns where the cost per unit length can be entered for each cost type, as shown in Figure 11.
4. Open the **Creating Libraries - Cost Template.xlsx** spreadsheet with the costs for this example. Navigate to the Pipe Costs tab, and copy the Material and Installation costs for the 1 inch to 36 inch pipe sizes into the worksheet that we just exported. Ensure that data is entered for material and installation for each of the sizes from 1 to 36 inch, and that the length units are listed as feet in the worksheet, since these were the length units used by the manufacturer for their pricing.
5. Save and close the worksheet.

Note: The length units used for the spreadsheet are taken from the default model units, which can be changed from User Options in the Tools menu.

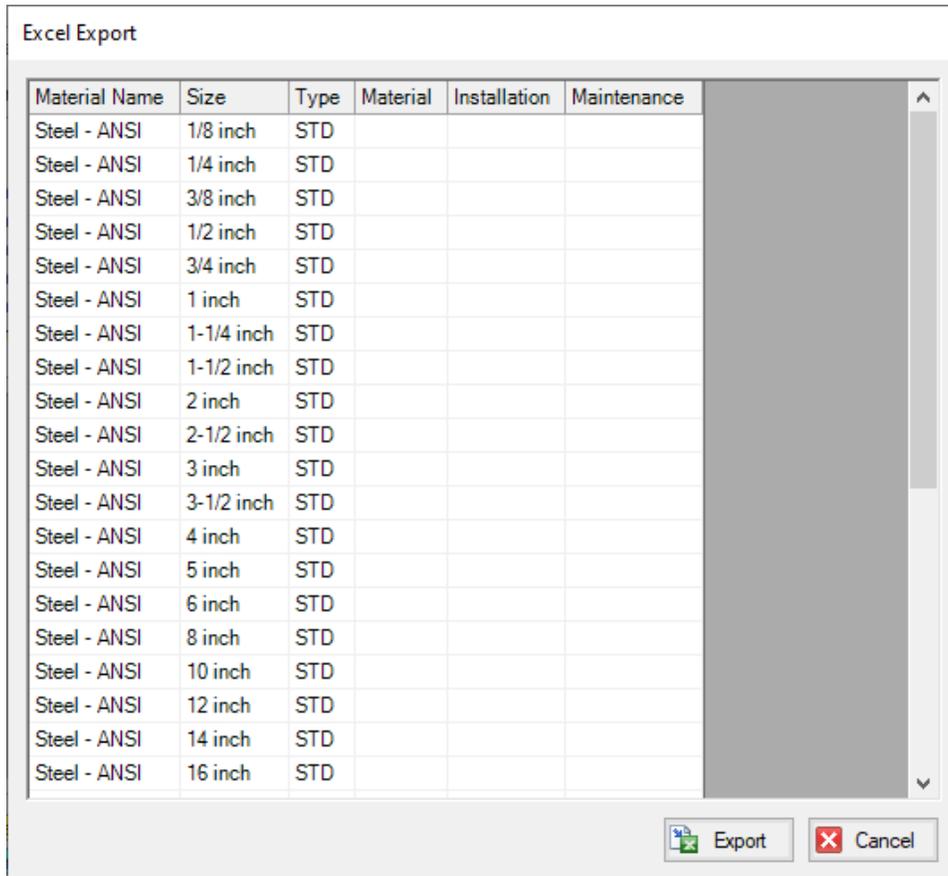


Figure 10: Excel Export window showing pipe items that will be included in the Excel sheet

	A	B	C	D	E	F
1						
2	Cost Units:	U.S. Dollars				
3	Length Units:	feet				
4				===== COST PER UNIT LENGTH =====		
5	Material Name	Size	Type	Material	Installation	Maintenance
6	Steel - ANSI	1/8 inch	STD			
7	Steel - ANSI	1/4 inch	STD			
8	Steel - ANSI	3/8 inch	STD			
9	Steel - ANSI	1/2 inch	STD			
10	Steel - ANSI	3/4 inch	STD			

Figure 11: Initial Excel sheet after exporting template for the STD Steel-ANSI pipe sizes

Back in Fathom in the Cost Library window, click the Import from Excel button, then browse to the worksheet we just updated and open it. When the Import has finished click **OK** on the notification from Fathom, then click the 1 inch pipe size under the Steel - ANSI STD pipe type. The Material and Installation costs should now be filled in, as shown in Figure 12. Click Save to save the cost library with a

unique file name **Steel - ANSI Pipe Costs.cst**, select Yes to make the library connected, then close the window.

Note: When using the Export/Import features for the cost library, it is not required to enter information for every size that was exported in order to import the worksheet. It is important to not delete any headings or descriptive information for the pipe data that was included in the template. Worksheets can be exported /imported as many times as desired, but whatever data is in the spreadsheet will overwrite anything existing in the library, meaning that if the costs are left blank for a size, any cost information will be deleted for that pipe size in the library. Due to this fact it is recommended to export the current information in the library first before importing any change information so that an accidental overwrite does not occur.

In the Library Manager, the new cost library should now be shown for the Steel - ANSI pipes, as shown in Figure 13, indicating that the pipe cost library was automatically connected.

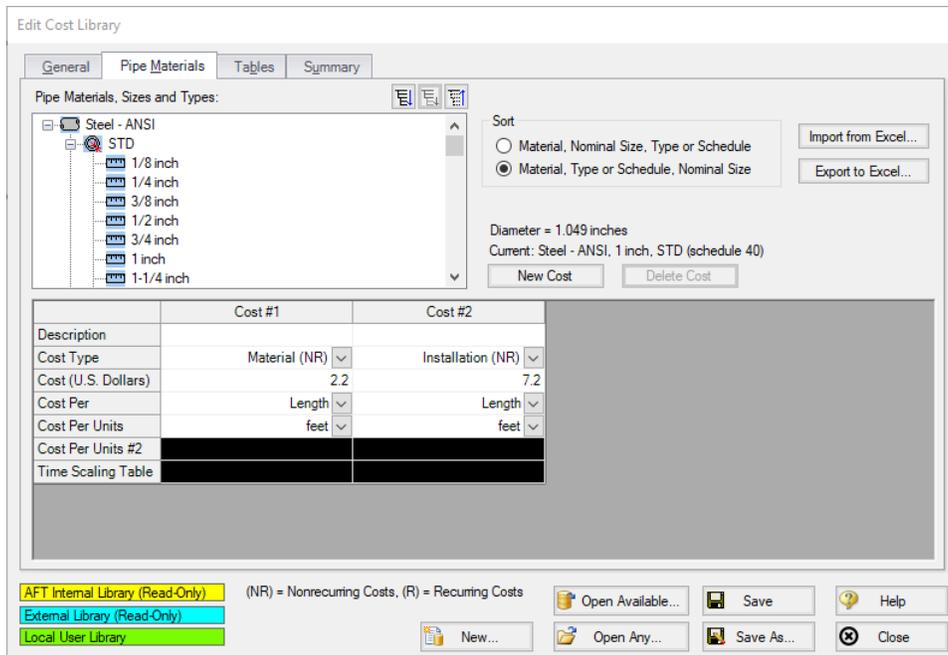


Figure 12: The Pipe Materials tab in the Cost Library window, showing the data recently imported

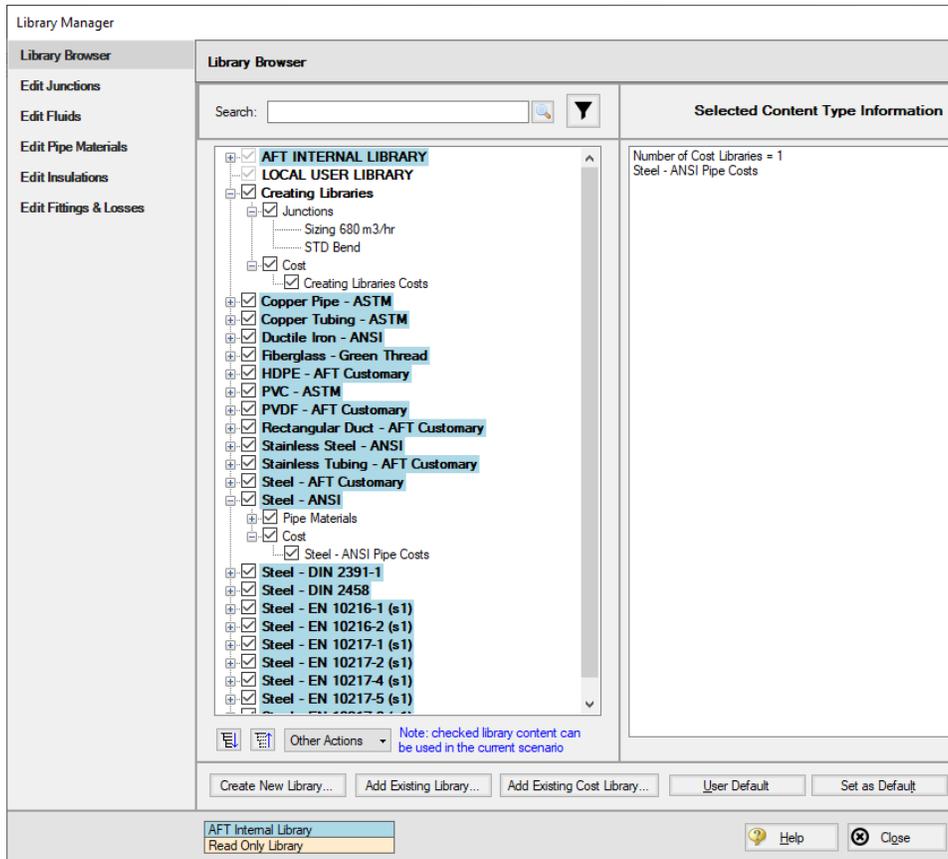


Figure 13: Library Manager after creating a engineering and cost libraries

Step 7. Define the Sizing window

To make use of the newly creating engineering and cost libraries, a few inputs are required in the **Sizing** window. Go to this window now.

A. Sizing Objective

By default, you should be on the **Sizing Objective** panel. For the Objective, select **Monetary Cost**. Then select **Calculate Cost, Do Not Size**.

B. Size/Cost Assignments

On the Size/Cost Assignments panel, make sure the **Pipes** button is selected at the top. Click **All** on the bottom left then click **Always Include in Cost** at the top to add all pipes to the Include in Cost list.

Click the **Pumps** button at the top to go to the pump size/cost assignments page. Select the radio button for both Pump 4 and 5 to **Include in Cost Report Only**.

Finally, go to the **General Junctions** button and add all four bends to the **Include in Cost Report Only** list.

C. Assign Cost Libraries

Go to the Assign Cost Libraries panel. **All 22 pipes** should assigned to the **Steel - ANSI Pipe Costs** library, as shown in Figure 14. Check the **Pumps** and **General Junctions** pages as well to confirm that those junctions are connected to the **Creating Libraries Costs**, as shown in Figure 15

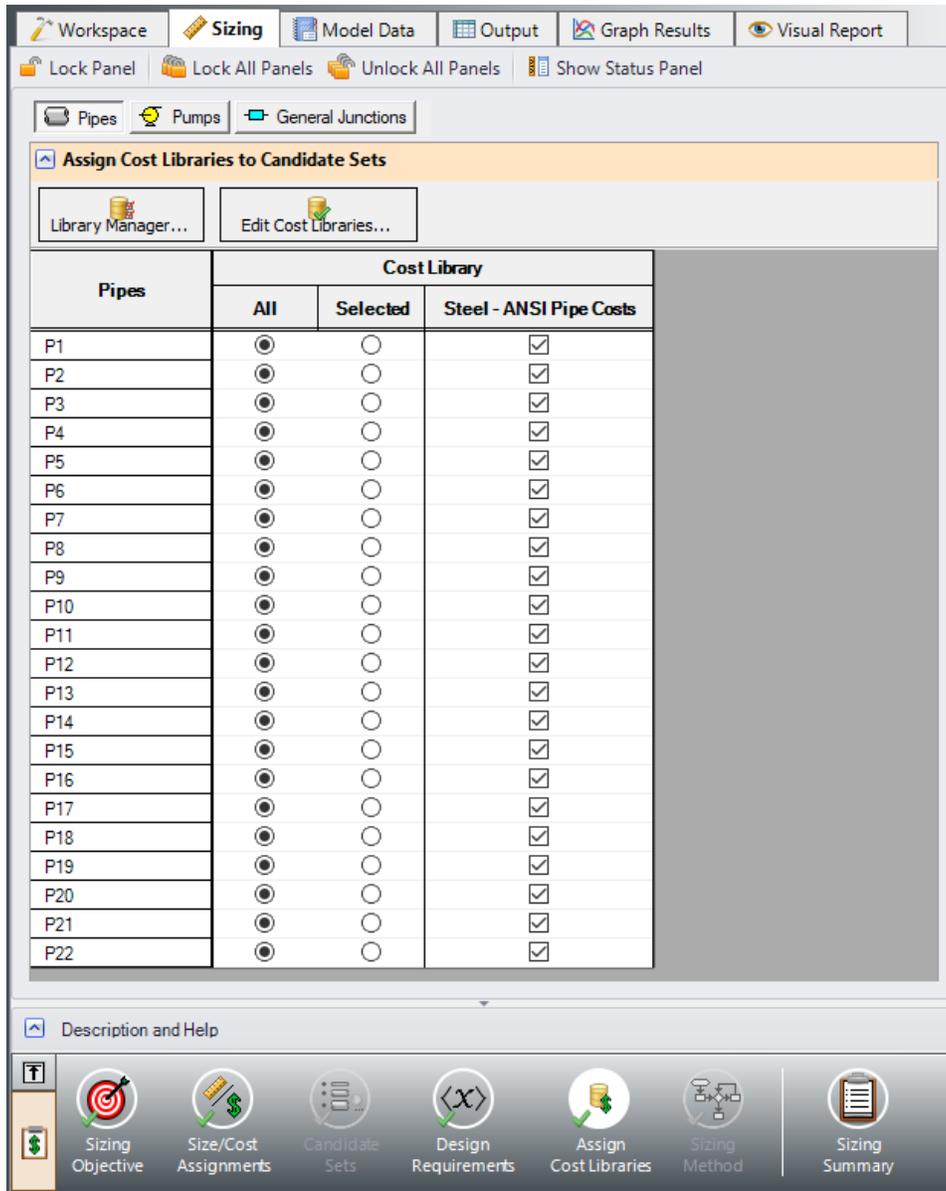


Figure 14: The Assign Cost Libraries panel with the new pipe cost library applied

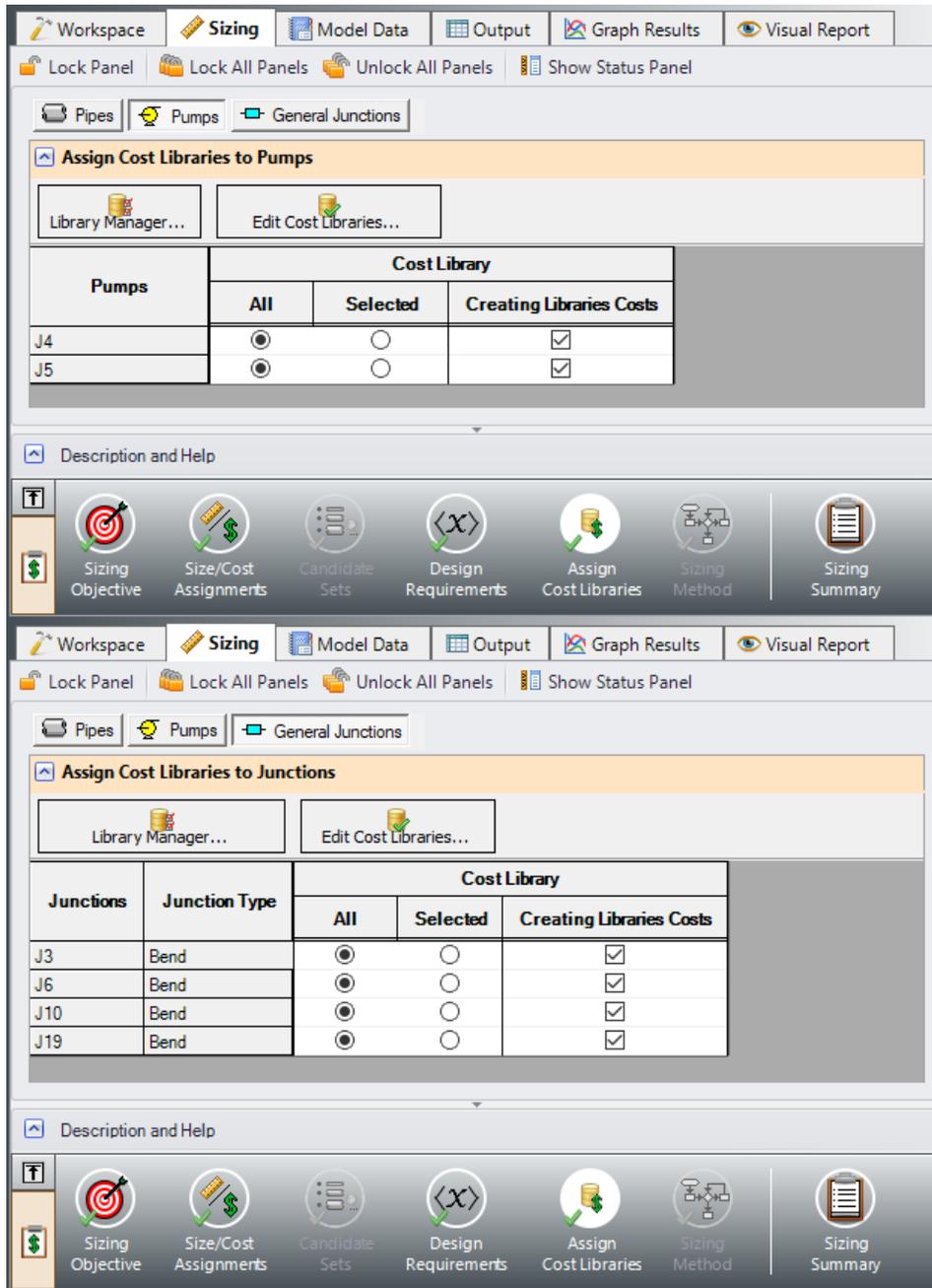


Figure 15: Applying the newly built cost library for the pumps and bends on the Assign Cost Libraries panel

In order to link our libraries to specific junctions in the model, we will need to apply the Engineering Library to the junctions in the Workspace. Open each of the **pump** junctions and select the **Sizing 680 m3/hr** item from the Library Jct list. Repeat this process for the four **bend** junctions in the model. Note that in this case the only thing that should change in the properties windows will be the name of the item from the Library list, since we created the library junctions based off of the junctions already defined in this model.

As a quick summary, we first connected the new libraries using the Library Manager. We then connected the junctions to the Engineering Library by using the Library Jct selection in each of the properties windows. In the Sizing tab we used the Assign Cost Libraries panel to select the corresponding Cost Library for each pipe and junction. The Assign Cost Libraries section is now complete.

Step 8. Run the Model and Check Output

Click Run from the toolbar or Analysis menu. Before the solver runs, the Model Diagnostics window will appear. Because there was no design requirements specified for the control valves, the control valves will add pressure if necessary. Typically, these messages should not be ignored. However, we will see from the output that the control valves will not add pressure. Therefore click Close on this message to dismiss it.

When the solver finishes, go to the Output window. On the Cost Report tab, the total cost should be \$35,261. In the Library Sources tab in the Junction section, each of the Bends and Pumps should list the Creating Libraries Costs as their library source as shown in Figure 16. The Library Sources in the Pipes section should similarly show the pipe cost library Steel - ANSI Pipe Costs for both Material and Installation.

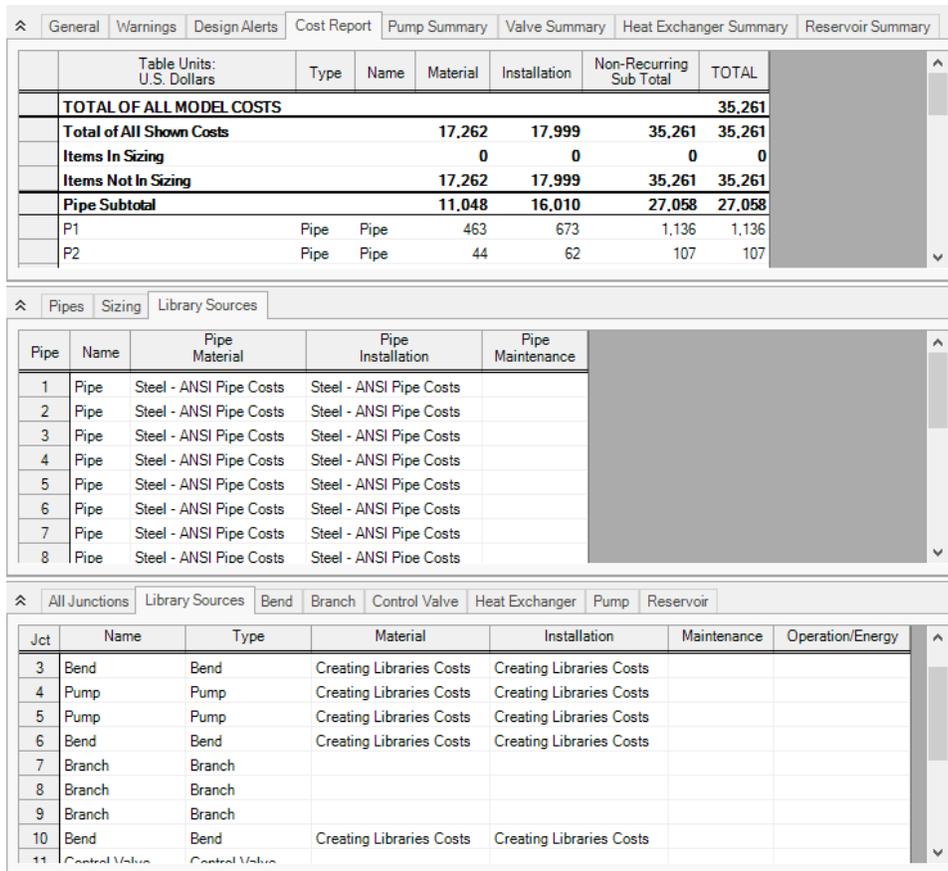


Figure 16: Output tab showing the current Library sources used in the model calculations

Cooling System - ANS

Summary

This example focuses on a closed loop cooling system that demonstrates some key features in using ANS to size a piping system. An existing model is used to investigate four potential sizing cases:

1. Size system for **initial cost** with a 10-year operating period
2. Size system for **life cycle cost** with a 10-year operating period
3. Size system for **life cycle cost** with an **initial cost limit** for a 10-year operating period
4. Size system for **life cycle cost** using a **manufacturer's pump curve** and with a 10-year operating period

The existing model was used to demonstrate AFT Fathom's capabilities for modeling closed-loop systems, pump curves, heat exchanger resistance curves, and fixed pressure drop control valves. This example will demonstrate how a system like the one built before can be seamlessly scaled up, meet design requirements, and minimize cost. For instance, the flow rate through the system will increase from 200 in the original model and system to 1,360 m³/hr. The pipes must be sized and their costs minimized accordingly.

Note: This example can only be run if you have a license for the ANS module.

Topics Covered

- Connecting and using existing engineering and cost libraries
- Sizing for initial or life cycle cost
- Using an initial cost limit with life cycle cost sizing
- Sizing a pump and sizing piping for the chosen pump
- Interpreting Cost Reports

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the [Beginner - Three-Reservoir - ANS](#) example, and is familiar with the basics of ANS analysis.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- **Metric - Cooling System.fth**
- **Metric - Cooling System - ANS.fth**
- **Cooling System.dat** - engineering library
- **Cooling System Costs.cst** - cost library for **Cooling System.dat**
- **Steel - ANSI Pipe Costs.cst** - cost library for **Steel - ANSI** pipes

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **Metric - Cooling System.fth** example file and save it as a new file. We will not need the existing child scenarios for this example: right click on the Base Scenario in the Scenario Manager and select **Delete All Children**.

Although the model is fully defined, a few alterations are necessary for this example to scale up the system. First make sure that the Workspace looks like Figure 1 below:

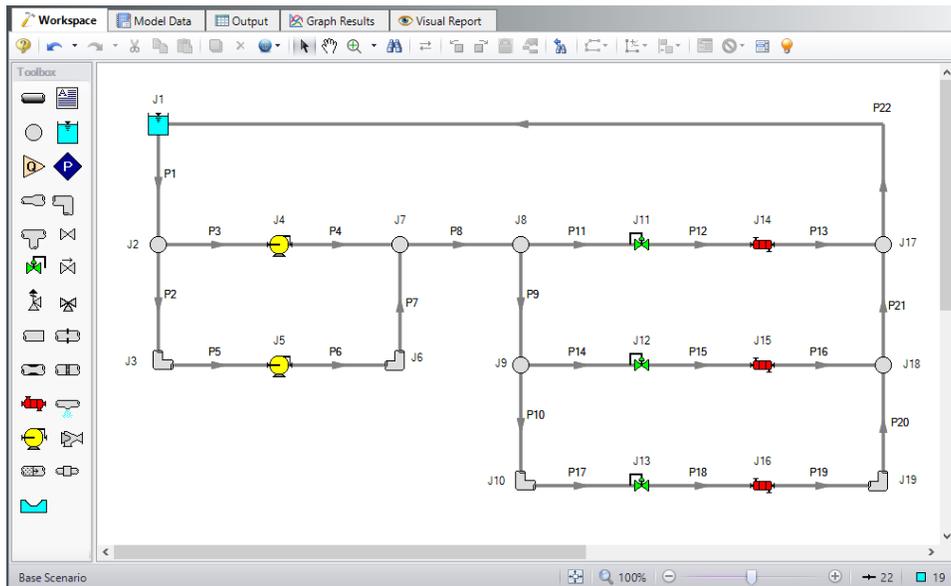


Figure 1: Workspace for Cooling System model

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate ANS and select Network to enable the ANS module for use. A new group will appear in Analysis Setup titled Automatic Sizing. Click OK to save the changes and exit Analysis Setup. A new Primary Window tab will appear between Workspace and Model Data titled Sizing. Open the Analysis menu to see the new option called Automatic Sizing. From here you can quickly toggle between Not Used mode (normal AFT Fathom) and Network (ANS mode).

Step 4. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = AFT Standard
 - b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
 - c. Temperature = 21 deg. C

Step 5. Define the Pipes and Junctions Group

As an initial guess to the scaled up system, we will change all pipe sizes to 36 inch. This should satisfy all Design Requirements and be a feasible solution, therefore it should be an acceptable initial guess for the system. Similar the junctions will be updated for the scaled up system.

Pipe Properties

Make the following changes to all pipes

1. Pipe Model tab
 - a. Size = 36 inch
 - b. Type = STD

Junction Properties

Make the following changes to these junctions.

1. J1 Reservoir
 - a. Liquid Surface Elevation = 9 meters
 - b. Liquid Surface Pressure = 0 barG (0 kPa (g))
 - c. Pipe Elevation = 0 meters
2. J11 & J12 Control Valves

- a. Valve Type = Flow Control (FCV)
 - b. Control Setpoint = Volumetric Flow Rate
 - c. Flow Setpoint = 450 m³/hr
 - d. Always Control (Never Fail) = Unchecked
3. J13 Control Valve
- a. Valve Type = Constant Pressure Drop (PDCV)
 - b. Control Setpoint = Pressure Loss
 - c. Pressure Drop = 0.345 bar (34.5 kPa)
4. J14-J16 Heat Exchangers
- a. Enter Curve Data =

Volumetric	Pressure
m ³ /hr	bar
0	0
450	3.45
900	13.8

Step 6. Overview of Sizing Systems with Pumps

A centrifugal pump can be modeled with either the Pump Curve or Sizing Analysis Type. Generally if a specific pump has been identified the Pump Curve type would be chosen, whereas the Sizing option is useful during the selection stage in order to identify the pumping requirements for the system. In some cases even though the specific pump has not been chosen several candidate pumps are available. For this third case it would be best to model each candidate pump as a pump curve using separate scenarios for each pump.

Sizing a Pump

The pumps in our cooling water case are being sized, so the Sizing Analysis Type was used with assigned flows of 680 m³/hr. In order to allow the pumps to be defined as fixed flows the flow control valve at J17 has been defined at a fixed pressure drop of 0.345 bar to ensure that the minimum pressure drop requirement for this valve is met during the sizing stage. This model thus represents a first cut model where, based on the results, the two pumps will become actual pumps with pump curves, and control valve J17 will become a flow control valve controlling to 680 m³/hr, similar to control valves J9 and J13.

As already stated, when a pump is modeled as an assigned flow, it is not a specific pump from a specific manufacturer. Thus, the costs for the pump can only be approximated. To a first approximation, it should be possible to estimate the non-recurring cost (i.e., material and installation cost) for the pump as a function of power requirements. For instance, a typical one horsepower pump may cost \$2000, and a ten horsepower may cost \$4000. Other typical costs for different power requirements can be approximated. The actual cost for the pump will, of course, highly depend on the application. Pumps for special

applications such as corrosive fluids or high temperatures may have different cost structures, but a reasonable approximation of cost for various power usage levels should be possible. These costs are entered into a cost library, and are accessed by the solver. As ANS evaluates different combinations of pipe sizes, each combination will require a certain power from the pump. With a cost assigned to this power, ANS can obtain a cost to enter into the objective function that it minimizes. Later in this example, we will look at the costs for the pumps.

In addition to non-recurring costs, recurring costs can also be estimated. Specifically, the cost of the power used by the pump over a period of time can be determined. All ANS needs is an overall pump efficiency to determine an actual power from the ideal power. Again, since we do not have a specific pump selected yet, the efficiency can only be approximated. ANS calls this the Nominal Efficiency. In addition, users can provide a Nominal NPSHR.

One may ask how important it is to choose appropriate values at this level. It depends. If the model only sizes for non-recurring costs, then the nominal efficiency will affect the actual power requirement of the pump, and hence the cost obtained from the cost library. If the cost of the pump is large compared to the cost of the piping, the specified nominal efficiency will likely play a significant role in the system selection.

In addition, for models that include energy costs in the objective, the energy cost may be a minor or major portion of the overall system cost. If it is a major portion, the results of the sizing will likely be very sensitive to the specified nominal efficiency.

Therefore, it is recognized that at this first cut phase the values for nominal efficiency and nominal NPSHR are imprecise and users may need to go through a few iterations to arrive at satisfactory results.

Modeling Pumps with a Pump Curve

When one is modeling a pump curve, accurate NPSHR and efficiency (or power) requirements can be specified for the pump. The cost data for that pump can also be accurately specified. During the automated sizing run the impact of the pump on the system sizing will be in the area of energy costs and in the requirement to satisfy the pump's design requirements.

First, let's discuss the issue of recurring cost. The non-recurring costs for the pump will be fixed, and will therefore not impact the sizing process. However, energy costs will vary with the power required for the pump. Thus including energy cost in the sizing will influence the pipe size selection.

Second, the specific pump has certain design requirements. For example, the sized network will be of no value if the NPSHR is violated. Thus, the pipes have to be selected by ANS in such a way as to satisfy all pump design requirements.

Since a specific pump is not input for our cooling water model (yet), no specific pump selection can happen at this point. It may frequently be the case that, based on the results of modeling the pump as an assigned flow pump, several candidate pumps may exist. The best approach is to create a scenario for each of these pumps, and rerun the sizing for each candidate pump. This will yield an ideal candidate pipe system for each candidate pump, from which an educated choice can be made for the system design. All else being equal, the selection would be the candidate pipe system that offers the lowest cost. When approached in this manner, one is selecting the pump and the piping system as an entire system, which offers improved matching of the pump to the system, and also offers significant opportunity for cost reduction.

Summarizing the Pump Selection Process

Based on the preceding discussion, we can outline a process for selecting a pump that offers a matched pump and piping system at the lowest possible cost. This discussion is an extension of the discussion in the Fathom Help file regarding sizing pumps and pipes concurrently. Here, we will augment that discussion with the considerations that are relevant to sizing.

1. Phase 1: Size the pump

- a. Model the pump (or pumps) as assigned flow pumps using the Sizing Analysis Type. If flow control valves exist, select the one that offers the lowest pressure drop margin (as discussed in the AFT Fathom Help file), and set that one to a pressure drop control valve at the minimum required pressure drop.
- b. The largest uncertainty at this point is the cost of the pumps. Based on actual pump cost data, specify the approximate cost of pumps of different power levels for different flow rates. This is entered into a cost library.
- c. Along with the design flow rate, specify a nominal efficiency that is consistent with the application and your experience. If you are unsure, take a guess and, after completing Phase 1, return to this step and improve your guess.
- d. If you have a reasonable idea of what the NPSHR will be, specify that in the nominal NPSHR. If you are unsure, ignore nominal NPSHR and proceed. After completing Phase 1, check the NPSHA and make sure that pumps are available at the head requirements obtained from the ANS results that have sufficient NPSHR margin.
- e. Determine whether the design goal is to minimize initial cost or life cycle cost, and select the appropriate objective in the Sizing Objective panel.
- f. Run the model and review the results. Take note of the relative cost difference between the piping and the pump(s). If the pump costs are a significant percentage, then continued iteration on Phase 1 will likely be worthwhile. Continued iteration would involve obtaining more precise cost information for pump(s) at the flow, head and power usage levels obtained from the Phase 1 results. It would also involve obtaining more precise information on nominal efficiency and nominal NPSHR.

2. Phase 2: Select a pump

- a. Phase 1 worked with approximate cost and engineering data for your pump. With that information ANS sized the network for an optimal pipe and pump system combination. At this point, that optimal system should be viewed as preliminary. Based on the pump operating point from Phase 1, identify some candidate pumps that match the requirements.
- b. Change any flow control valves set to pressure drop control valves (as specified in Phase 1a) back to flow control valves.
- c. Identify the operational requirements for the flow control valves, such as minimum pressure drop, and enter those as Control Valve Design Requirements.
- d. For each candidate pump, create a child scenario. Each child scenario should be created below the scenario that has the flow control valves as discussed in Phase 2a and 2b.
- e. Enter the pump head, NPSHR and efficiency (or power) curve. Enter the actual pump cost as well into a cost library.
- f. Run each scenario. All else being equal, the scenario with lowest cost will be the preferred design. The pipe sizes obtained from this final solution may differ slightly from those obtained Phase 1. If there are significant pipe size differences between Phase 1 and Phase 2, it may indicate that the actual pump was not matched very well to the Phase 1 requirements. It could also mean that there was an error somewhere in the process.

- g. In Phase 2f, it may not be true that all else is equal. For example, even if one design offers lower cost, you may prefer a higher cost system because your experience suggests the pump is more reliable. On another note, if it is possible to quantify the cost reliability, this can be included in the cost of the junction and the automated sizing process.

For this example, we will be completing the pump selection process above considering the objective of minimizing initial cost, then minimizing life cycle cost.

Step 7. Configure Sizing Settings

Load the Base Scenario. The hydraulic model is already defined for a regular Fathom run, but we need to complete the sizing settings before running the analysis. Navigate to the Sizing window by clicking the Sizing tab at the top.

A. Sizing Objective

The Sizing Objective window should be selected by default from the Sizing Navigation panel along the bottom. For this analysis, we are interested in sizing the system considering the monetary cost for the initial costs only.

1. Sizing Option = Perform Sizing
2. Objective = Monetary Cost
3. Option = Minimize
4. Options to Minimize Monetary Cost = Size for Initial Cost, Show Energy Costs
 - This will update the cost table to include both initial costs in the sizing calculation, and moves the Energy costs to be included in the Cost Report so that we can later compare the costs in this scenario to one where the Energy costs are included for the sizing. We do not have any information on Maintenance cost, so we will leave this cost in Ignore Cost.
5. System Life = 10 years
6. Energy Cost = 0.11 U.S. Dollars Per kW-hr
 - Note this assumes the pump will run all day, every day, for the entire 10 years. Alternatively, we could have created an Energy Cost Library, which would be connected in the Assign Cost Libraries section. An Energy Cost Library allows variable energy costs to be used to account for varying prices during different times of day, different seasons, etc. For now, we are simplifying the input by using a fixed cost.

The Sizing Objective window should now appear as shown in Figure 2.

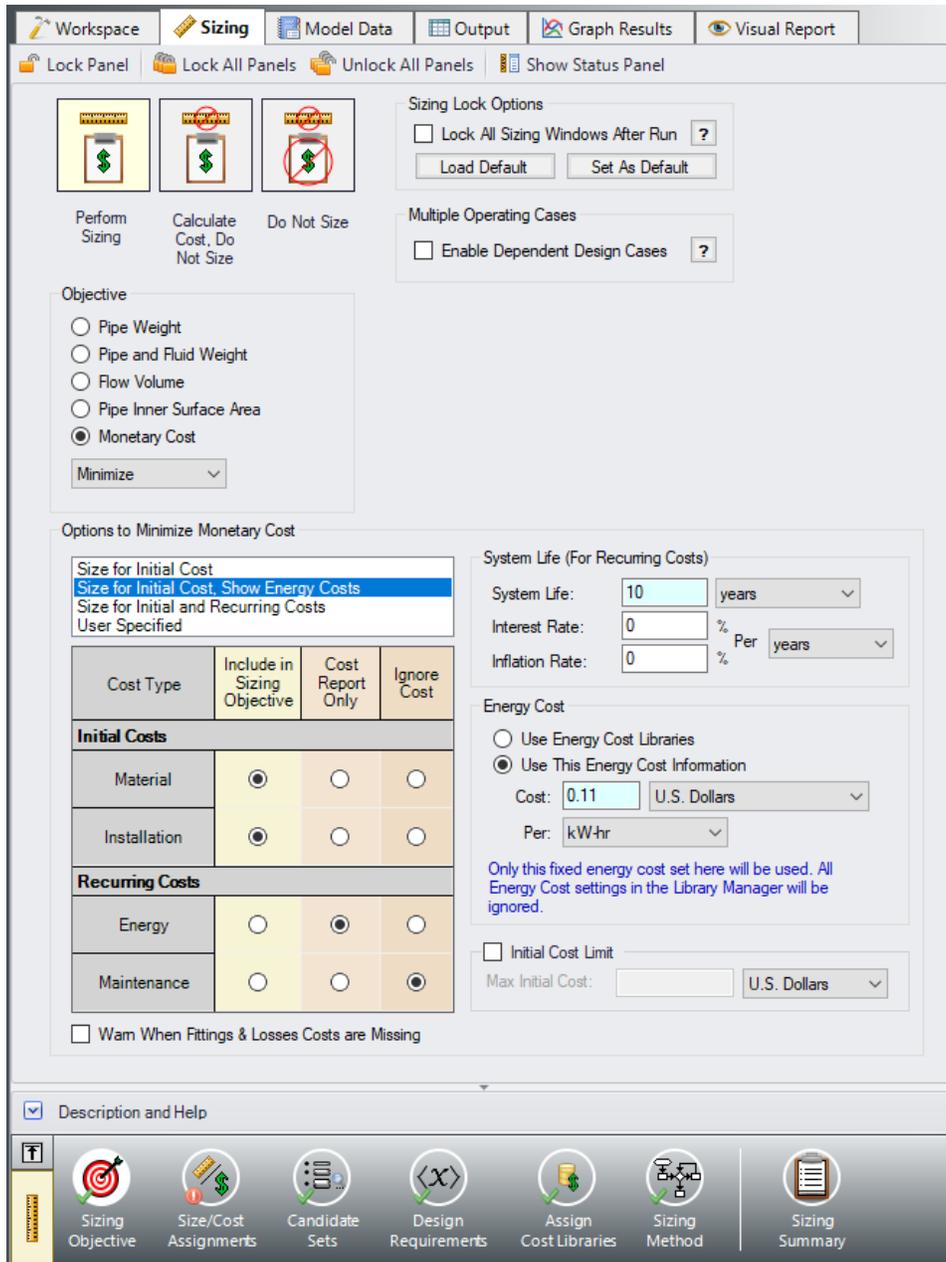


Figure 2: Sizing Objective panel setup for the initial cost analysis

B. Size/Cost Assignments

On the Sizing Navigation panel select the Size/Cost Assignments button. For this example we are sizing both the pipes and pumps for a new system, so we will need to calculate and minimize cost for all related objects in the model.

➤ **Click All under the table** to select all of the pipes, then click the Always Include in Cost button next to Move to. This will move all of the pipe selections to the corresponding column to be sized.

To decrease the run time and reduce complexity, as well as to enforce some design uniformity, it is useful to create Common Size Groups to link the pipes in different sections of the model.

For example, if all the 22 pipes are selected to be Not In Group, then 22 unique pipe sizes will be selected by the automatic sizing engine. That could potentially mean, for example, different suction pipe diameters in pipes 1 and 2. A similar thing would happen for discharge pipe diameters. If one wanted the suction pipes 1 and 2 to be sized to the same diameter, one would group them into a Common Size Group. To see how this works, in this model all suction and discharge piping for both pumps will be put into a Common Size Group. What this means is that there will be one unique diameter chosen for this group of pipes, rather than potentially six different diameters.

For this system we will define five groups to be sized in this model. To set the groups, it is standard to select pipes which must have the same size due to their placement relative to other pipes and junctions, such as pipes at the entrance and exit of valves. In Figure 3 we have identified the five sections of the model to group the pipes.

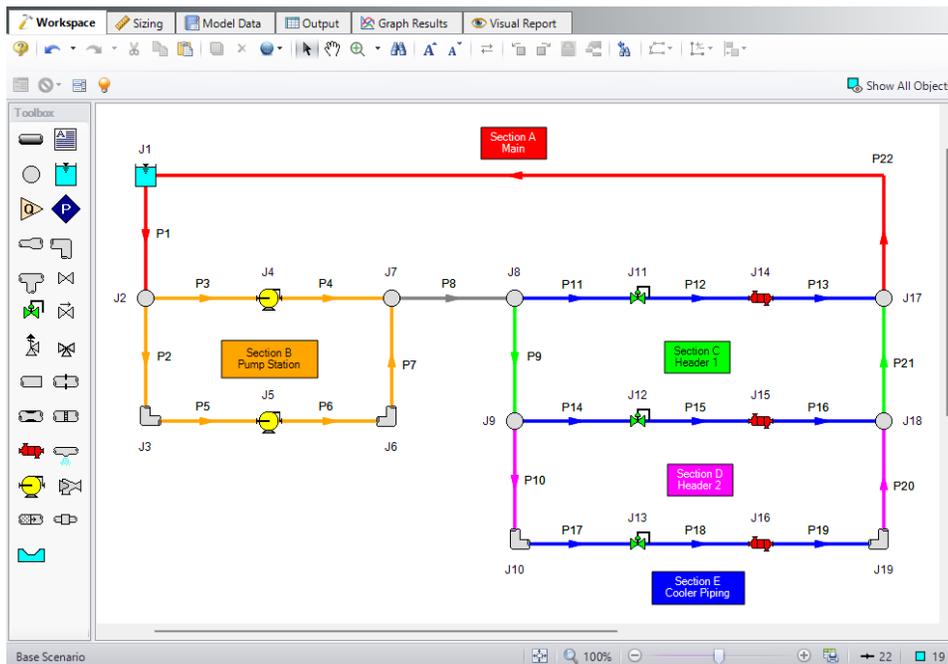


Figure 3: Workspace with separate sections identified to create Common Size Groups

First let's create a *Main* size group for pipes 1 and 22, which are red in Section A. We would expect these lines to be sized the same, as the inlet and outlet for the cooling reservoir.

Note: If desired, the coloring and annotations in Figure 3 and in the completed model file can be recreated using [Workspace Layers](#).

➤ **Click New above the Pipe Grouping section** to create the group, and name it *Main*. Select the radio buttons under *Main* next to 1 and 22 to add them to the group. Alternatively, both pipes could be selected

in the Workspace, then be added to a new group by right-clicking the Workspace and choosing Add Pipe (s) to Common Size Group.

Now let's repeat this process for the piping in Section B around the pumps and control valves. Since this section contains identical pumps operating in parallel, we would expect the pipes in this section to have a common size. Similar to above, create a new group called *Pump Station*, and add pipes 2-7 as shown in orange in Figure 3 in section B to the new group by selecting them in the Workspace, or using the radio buttons.

Next, let's consider Section C-E, the piping near the coolers. In this case we have three coolers in parallel, with several segments of pipe connecting the cooler piping segments. We would expect the pipes directly connected to the coolers to have a common size, but the header piping (9, 10, 20, 21) would likely need to be larger, so we will group them separately. Create three Common Size Groups as follows:

1. *Header 1* with 9 and 21 (Section C, lime green)
2. *Header 2* with 10 and 20 (Section D, magenta)
3. *Cooler Piping* with 11-19 (Section E, blue)

The completed Size/Cost Assignments can be seen in Figure 4. Note that pipe 8 should be the only pipe not included in a group. This leaves six different pipe sizes that the ANS module will independently solve for during the automated sizing, one for each of the groups, and one for pipe 8. This is indicated in the bottom right of the Size/Cost Assignments panel, which shows that six independent pipe sizes exist for this configuration.

Note: Pipes in the Size/Cost Assignments table can be sorted by Common Size Group instead of sorting by number by clicking on the Pipe Grouping header. Clicking on the header for a particular group will sort the pipes in that specific group to the top of the window.

Pipes	Sizing and Cost Options					Pipe Grouping				
	Automatically Size		Do Not Size		Not in Group	Common Size Groups				
	Always Include in Cost	Include in Cost if Size Changes	Include in Cost	Do Not Include in Cost		Main	Pump Station	Header 1	Header 2	Cooler Piping
P1	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
P2	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
P3	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
P4	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
P5	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
P6	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
P7	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
P8	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
P9	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>
P10	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>
P11	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
P12	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
P13	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
P14	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
P15	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
P16	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
P17	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
P18	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
P19	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
P20	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>
P21	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>
P22	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>

Figure 4: Size/Cost Assignments panel with Common Size Groups defined as outlined above

➤ **Click the Pumps button** to switch to view the Pump Size and Cost Assignments. Select the option to Include in Cost Report and Sizing for both pumps so that the ANS module will size the pumps along with the pipes as shown in Figure 5.

Similar to the Common Size Groups for pipes, Pumps can be placed into Maximum Cost Groups based on monetary cost. If pump costs are added to a Maximum Cost Group, the ANS module will determine the pump in the group with the maximum costs, and set all pumps in the group to use the largest cost. In this case we will allow the pumps to be sized independently, so no groups need to be created.

We also want to include the Bends in the sizing for this model.

➤ **Click the General Junctions button** and change each of the Bends (3, 6, 10, and 19) to Include in Cost Report and Sizing (Figure 6)

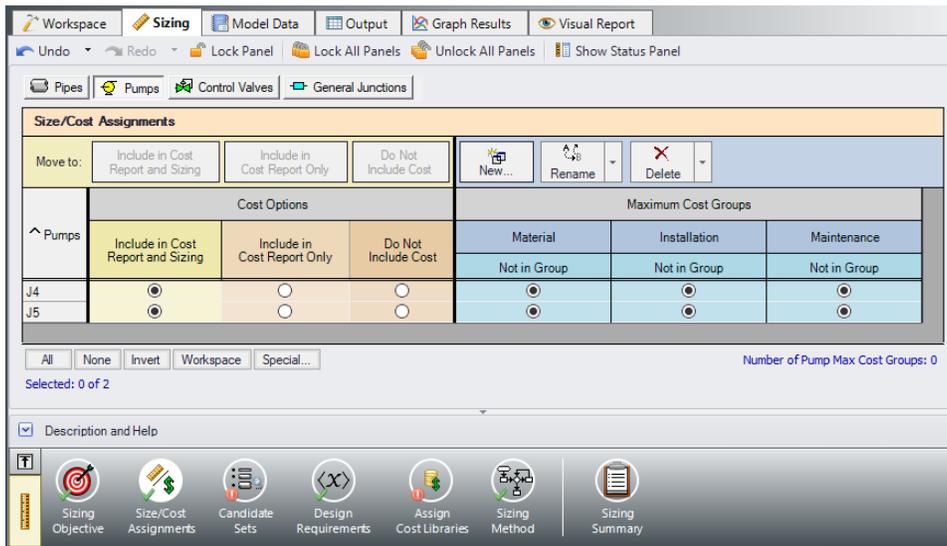


Figure 5: Size/Cost Assignments panel defined for the pumps

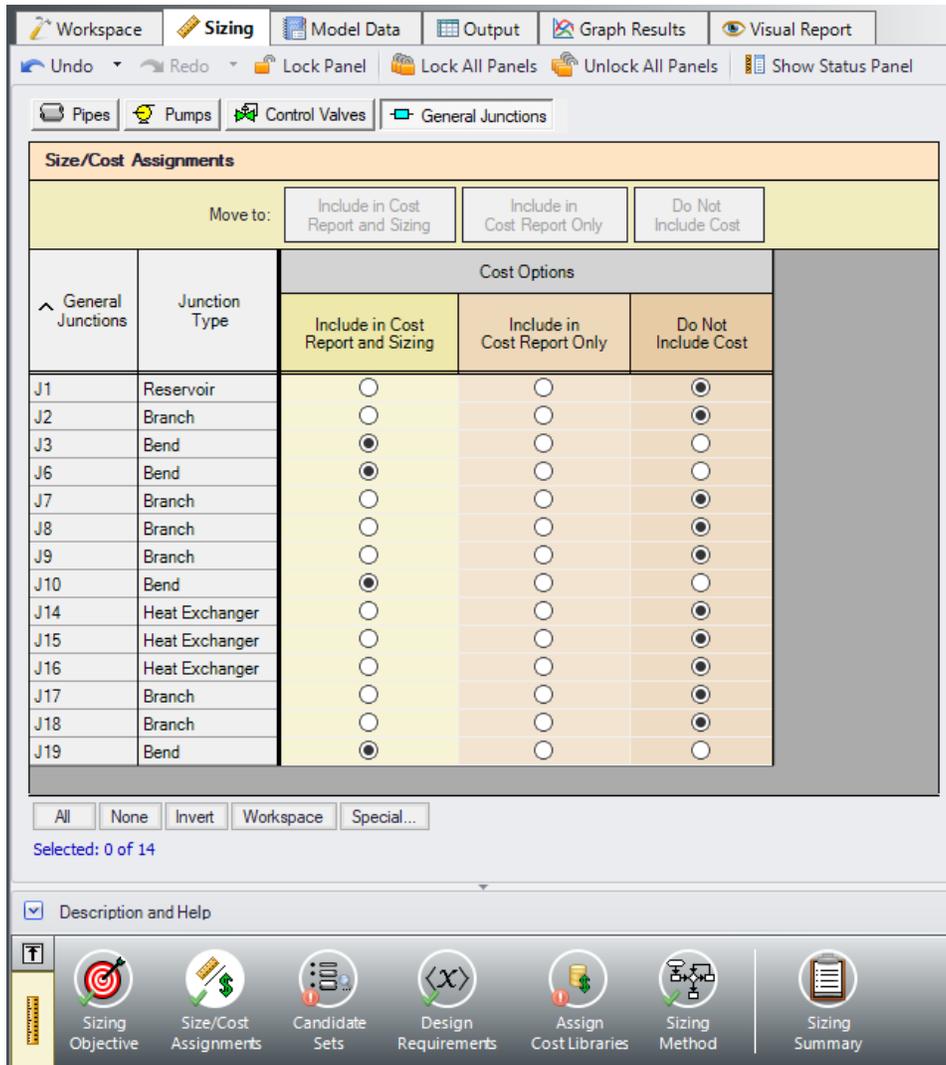


Figure 6: Size/Cost Assignments panel defined for the General Junctions

C. Candidate Sets

Click on the Candidate Sets button to open the Candidate Sets panel.

Click New under Define Candidate Sets and name it *STD Steel 1"-36"*. In the Select Pipe Sizes window, select Steel - ANSI from the dropdown menu, expand the STD group (make sure the Sort selection on the bottom is Type, Schedule, Class), then add all pipes from 1 inch to 36 inch to the Candidate Set list on the right. Click OK to close the Select Pipe Sizes window.

In the bottom section of the window, assign each of the Common Size Groups and pipe P8 to the STD Steel 1"-36" Candidate Set. The window should appear as shown in the figure below.

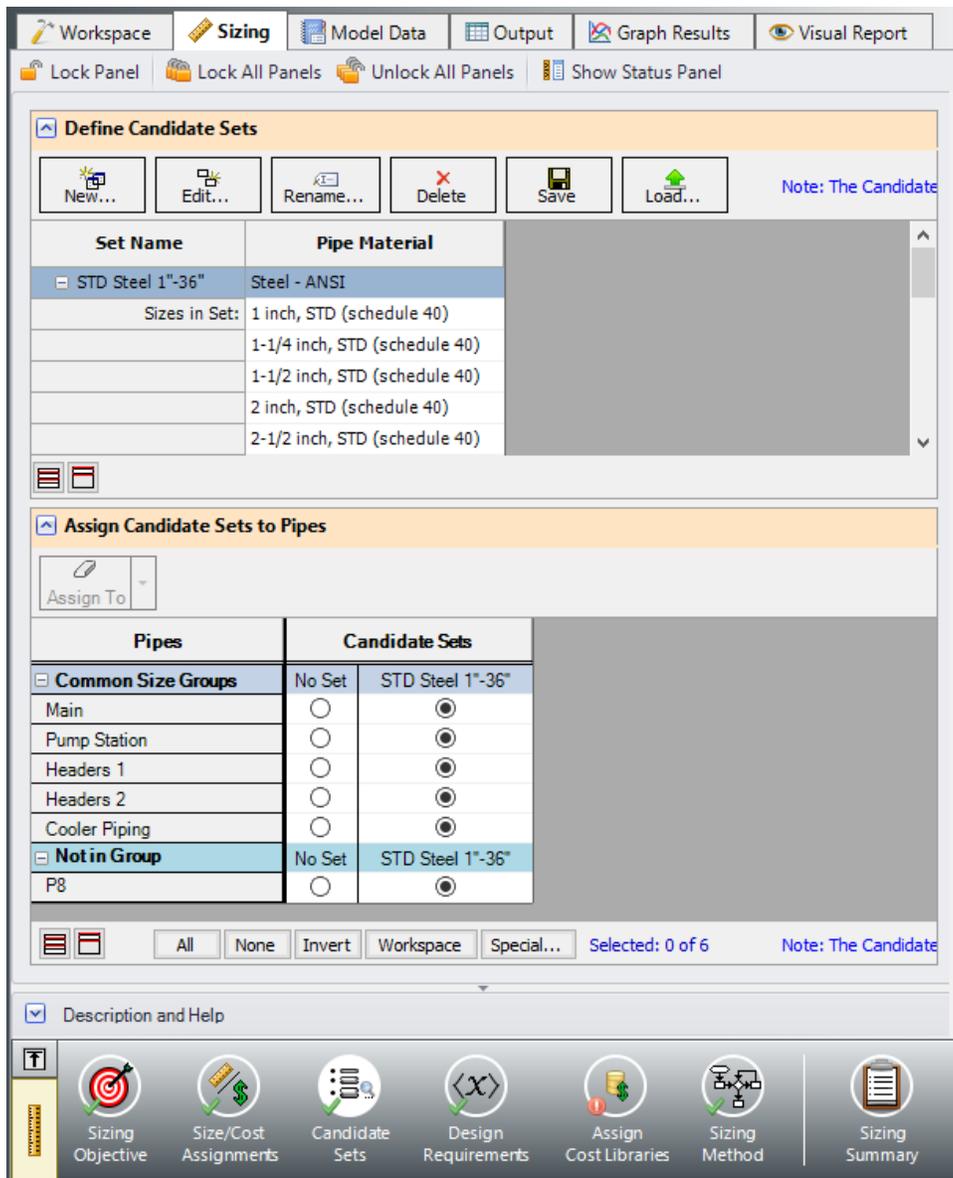


Figure 7: Candidate Sets panel fully defined for the model

D. Design Requirements

Select the Design Requirements button from the Sizing Navigation Panel.

For this model we have multiple requirements for the system:

1. *Max Velocity*: All pipes have a maximum velocity of 3.66 meters/sec
2. *Min Pressure*: All pipes have a minimum static pressure requirement of 1 bar

3. *NPSH Margin*: All pumps must have a minimum NPSH Margin of 3 meters (for this system the nominal NPSHR is 15 meters)
4. *Design DP*: All control valves must have a minimum static pressure drop of 0.345 bar
5. All heat exchangers must receive a minimum flow of 430 m³/hr

Define and assign the first two requirements listed above as Pipe Design Requirements as shown in Figure 8, and item 3 as a Pump Design Requirement as shown in Figure 9. Notice that we have accounted for the minimum flow requirement in this system by using flow control valves at each of the heat exchangers. By using assigned flow pumps and flow control valves at the heat exchangers (we will do this later) we will incorporate the necessary flow rate as a boundary condition, so we do not need to apply a Design Requirement to account for this item.

For Control Valves 11 and 12 define and apply the minimum pressure drop requirement in the Control Valves section as shown in Figure 10. Note that for pump sizing purposes as discussed earlier Control Valve 13 has been defined as a Constant Pressure Drop valve, so the pressure drop Design Requirement does not need to be applied to this valve.

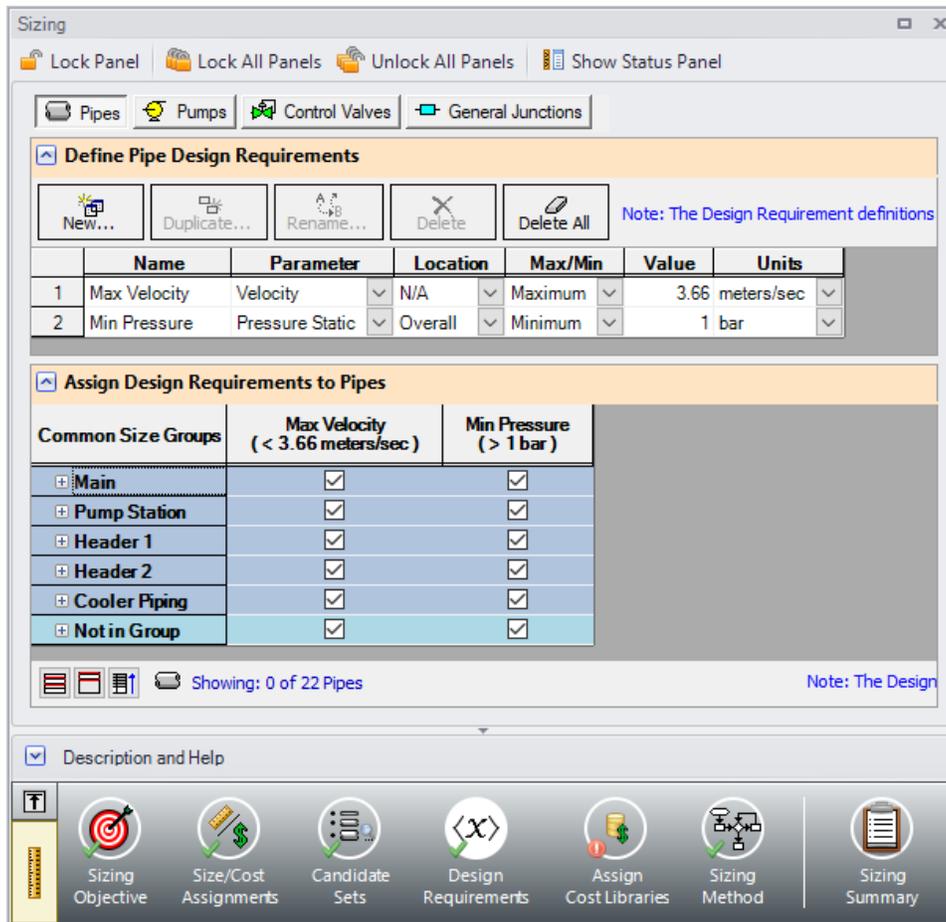


Figure 8: Design Requirements panel for pipes

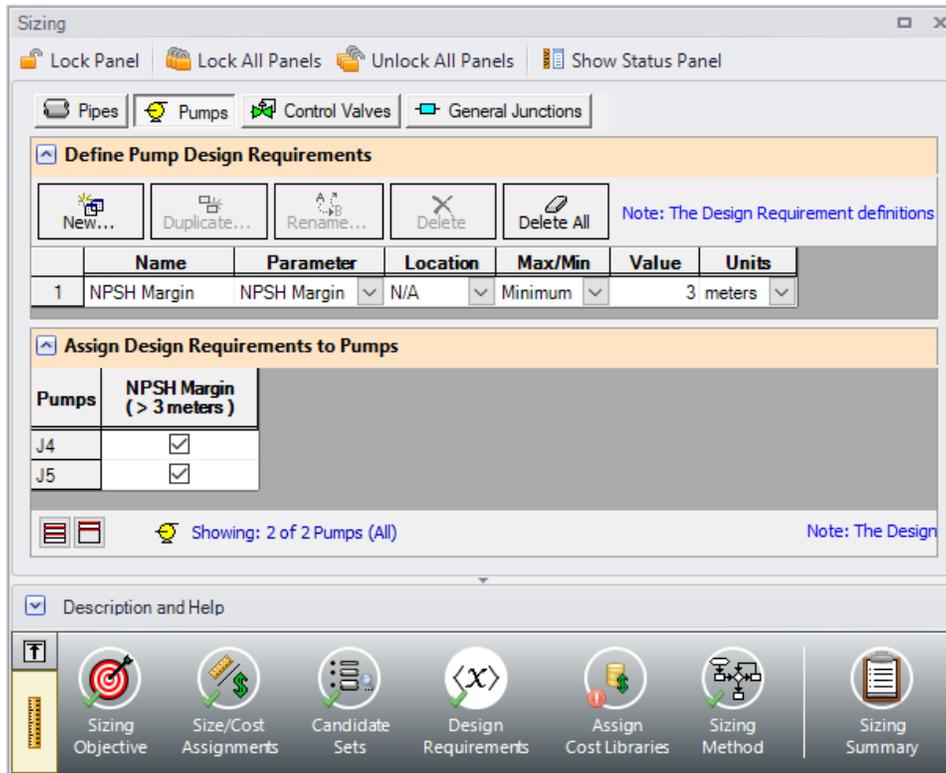


Figure 9: Design Requirements panel defined for the pumps

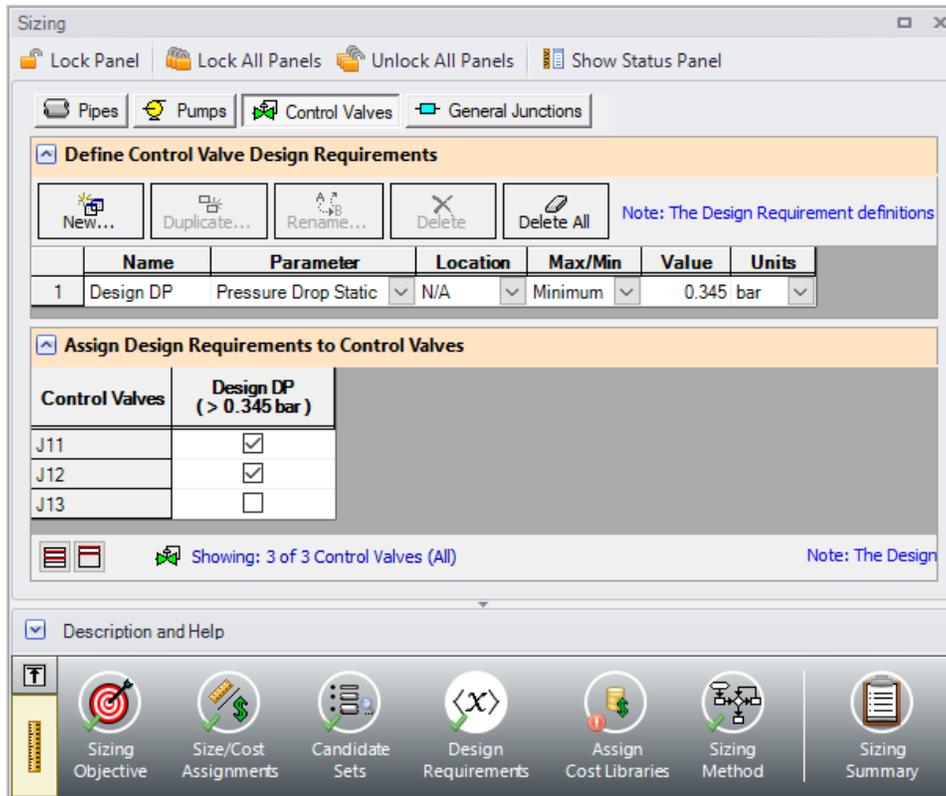


Figure 10: Design Requirements panel defined for the control valves

E. Assign Cost Libraries

Select the Assign Cost Libraries button. For this model the engineering and cost libraries have already been created, but we will need to connect and apply them.

Note: The [Cooling System, Creating Libraries - ANS](#) example provides instructions on how to build the libraries for the first part of this example, and may be completed later for practice with creating libraries. The provided pre-built libraries will be needed to obtain the pump curve data for Step 16 of this example.

➤ **Open the Library Manager** by opening the Library menu and selecting Library Manager. The AFT INTERNAL LIBRARY, LOCAL USER LIBRARY, and the pipe material libraries should be available and connected by default. Available libraries will be shown based on libraries that Fathom has identified on your machine. The default connected libraries will be visible here, and additional libraries may be visible from other example files or from libraries previously built and used on your machine.

This example uses one user-defined engineering library **Cooling System.dat**, which contains the data for the pumps and bends. There are also two cost libraries, **Cooling System Costs.cst** and **Steel - ANSI Pipe Costs.cst**, which contain the cost data for the junctions and the pipe materials, respectively. **Cooling System Costs.cst** was built off of the engineering library **Cooling System.dat**, and **Steel -**

ANSI Pipe Costs.cst was built off of the Steel - ANSI Pipe Material Library. To connect these libraries, do the following:

1. Click Add Existing Library.
2. Browse to the Fathom 13 Examples folder (located by default in C:\AFT Products\AFT Fathom 13\Examples), and open the file titled **Cooling System.dat**.
3. Repeat the above steps, but choose Add Existing Cost Library to browse for **Cooling System Costs.cst**, then **Steel - ANSI Pipe Costs.cst**.

Note that it may be possible that the model file already has the cooling system libraries connected, in which case you will be notified that the library you are trying to add is already available. If the libraries are available, you will simply need to make sure they are connected by ensuring a checkmark is next to the name. There should be a library titled **Cooling System** with two sub libraries: **Junctions** and **Cost** (Cost should have a cost file titled **Cooling System Costs** beneath it. Additionally a Cost sub library should appear subordinate to the Steel - ANSI, the library **Steel - ANSI Pipe Costs** should appear beneath it.

Note: When a library is selected in the Library Manager, the filename and location can be viewed in the Selected Content Information section.

Once you have confirmed that the necessary libraries are connected, click Close to exit the Library Manager.

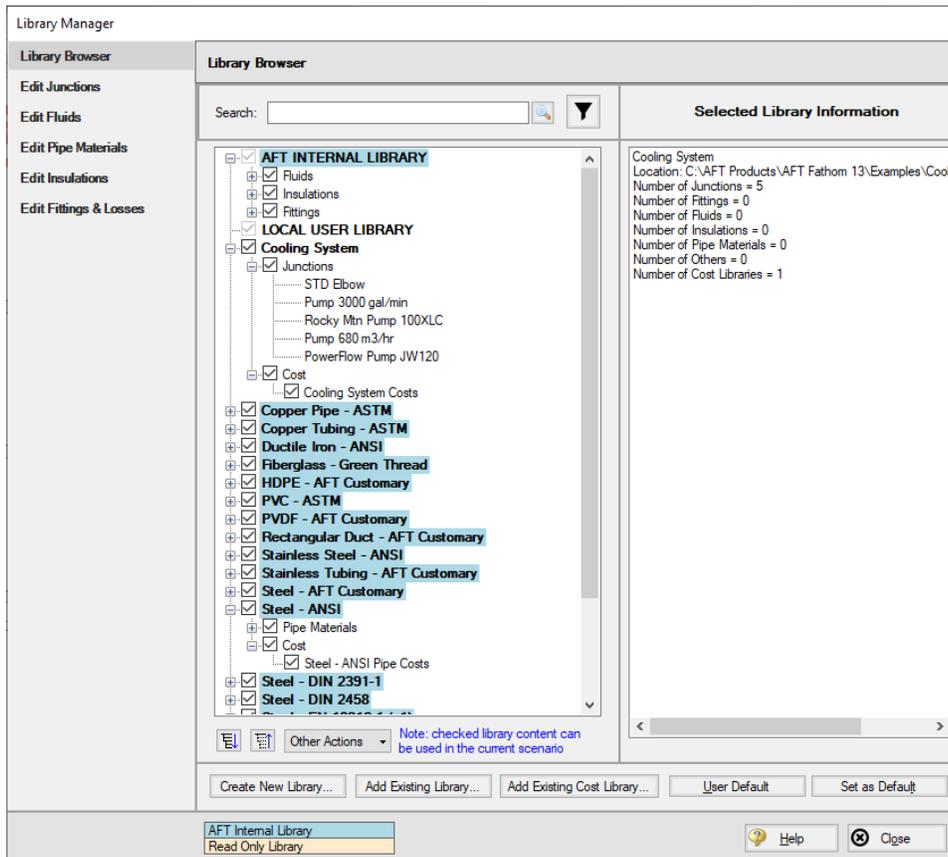


Figure 11: Library Manager with Cooling System junction libraries connected

Since we are connecting a pre-built engineering library, we will need to take an additional step to confirm that the junctions in the Workspace are linked to the Cooling System engineering library that we added.

1. Go to the Workspace, and open Pump J4
2. From the Library Jct list, select **Pump 680 m3/hr**. The pump input should change, and Fathom will recognize that this junction is connected to the engineering and cost library for this pump. Click **OK**.
3. Repeat step 2 for Pump J5.
4. Open each of the Bends (junctions 3, 6, 10, and 19), and select **STD Elbow** for each from the Library Jct list, and click **OK**.

We have now re-connected the junctions that will be considered for the Cost Report and Sizing Calculations to the engineering library. This is important, since a junction must be connected to an engineering library before a cost can be assigned for it. Let's now return to the Sizing tab.

Back in the Assign Cost Libraries panel for the Pipes, All should be selected for the STD Steel 1"-36" inch Candidate Set with only the Steel - ANSI Pipe Costs library shown. The user can create and connect as many cost libraries as desired, which can be useful to easily update the cost of individual objects, rather than having to keep all of the costs in one large library. If multiple cost libraries are assigned to a Candidate Set, then the costs will be stacked to find a cumulative cost.

➤ **Click on the Pumps button** to view the cost library assignments for the Pumps. The only visible library should be the Cooling System Costs library, as this was the only cost library connected for the associated engineering library. Make sure that the checkboxes are selected under the library name so that the cost data will be applied for each of the pumps. To review the cost data for the Bends click the General Junctions button to verify that the Cooling System Costs are being applied to each of the bends.

F. Sizing Method

Select the Sizing Method button to go to the Sizing Method panel.

➤ **Choose Discrete Sizing**, if not already selected, since it is desired to select discrete sizes for each of the pipes in the model.

For this model we have six independent pipe sizes as mentioned earlier (five Common Size Groups and pipe 8), and 48 design requirements (2 pipe design requirements applied to all of the pipes, the pump NPSH requirement, and the minimum pressure drop for control valves). Due to the number of independently sized pipes, it would be recommended to try the MMFD or SQP search method.

Select **Sequential Quadratic Programming (SQP)** for the sizing method.

Note: [Search Methods](#) can give different results in certain cases. It may be prudent to try several methods and to proceed with the one that yields optimal results.

G. Sizing Summary

From the Sizing Navigation Panel select the Sizing Summary button.

The Sizing Summary panel allows the user to view all of the sizing input for the model in a tree view layout, which is shown in Figure 12. Items can be organized by sizing parameters (Design Requirements,

Candidate Sets, etc.), or information can be viewed based off an individual pipe. For clarity these categories can be shown/hidden using the check boxes at the top of the panel.

Each tree node can be expanded/collapsed individually by using the '+' and '-' buttons on the node. Alternatively, the buttons on the bottom of the panel can be used to collapse/expand all items. Right-clicking on the nodes provides additional options to collapse/expand items, view related nodes, or to copy information to the clipboard. The Sizing Summary panel can be printed from the printer button at the bottom of the panel.

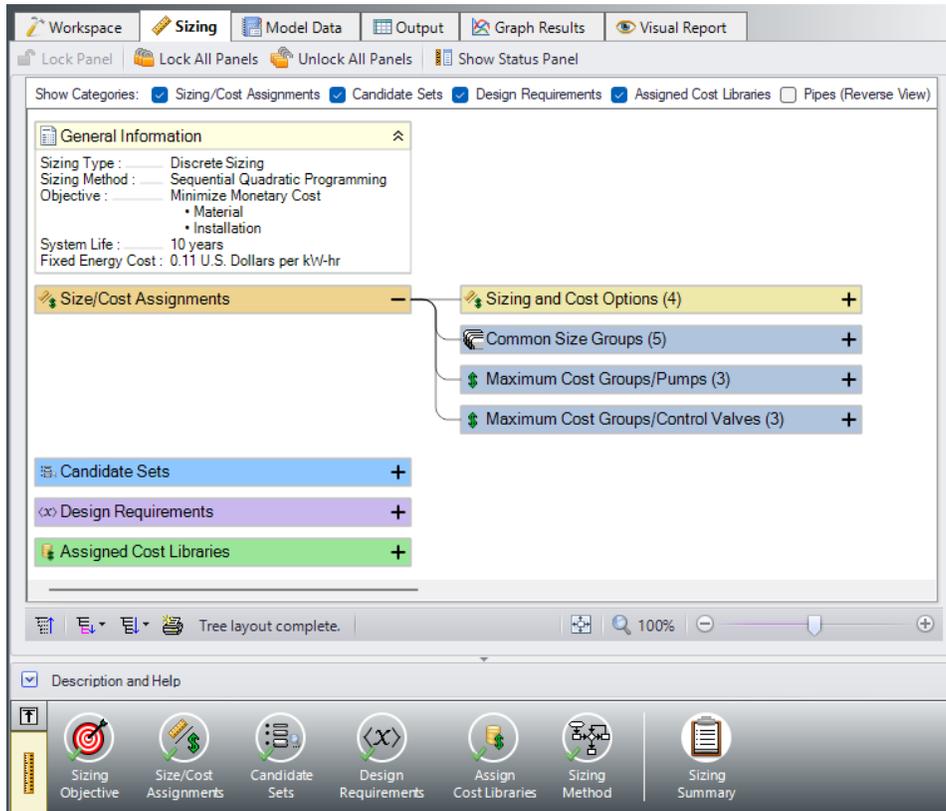


Figure 12: Sizing Summary panel layout with the Size/Cost Assignments expanded. The number of children nodes for each item can be seen in parentheses.

Step 8. Set up the Scenario Manager

To compare the four potential sizing cases, child scenarios will be set up in the Scenario Manager. Go to the Scenario Manager now and create the following child scenarios based on Figure 13 below. These will include the Not Sized scenario and the four potential sizing cases: Initial Cost, Life Cycle Cost, Initial Cost Limit, and Actual Pump Curve.

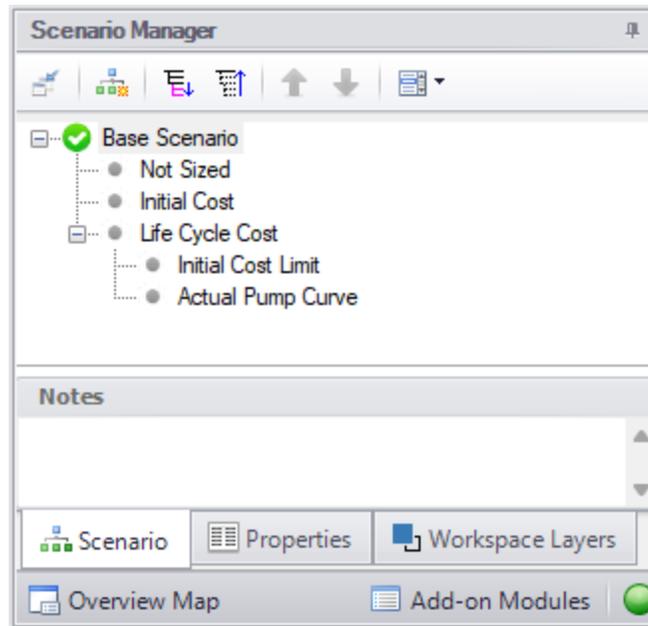


Figure 13: Scenario Manager showing the child cases for the Cooling System ANS example

Step 9. Set up the Not Sized scenario

Load the **Not Sized** scenario by double-clicking on it in the Scenario Manager.

Navigate to the **Sizing** window and go to the **Sizing Objective** panel. Select **Calculate Cost, Do Not Size**. In the Options to Minimize Monetary Cost, the selection should be User Specified. Make sure that the radio buttons for **Material**, **Installation**, and **Energy** are selected for **Cost Report Only**, while **Maintenance** should have **Ignore Cost** selected.

This will calculate the life cycle cost of the system before any sizing is performed. This cost will be used as a comparison to the sized scenarios.

Step 10. Run the Model

Select Run Model in the Analysis menu or click Run Model in the Common Toolbar. This will open the Solution Progress window. This window allows you to watch as the Fathom Solver and ANS module converge on a solution.

After the run has been completed, the results can be reviewed clicking the Output button at the bottom of the solution progress window.

Step 11. Review the Not Sized results

The Cost Report is shown in the General Section of the Output window (see Figure 14). No sizing was performed, so all cost values will appear in the Items Not In Sizing row. The total cost for this system is \$2,491,404. This includes all costs over 10 years. The initial cost is the total of the material and installation cost: \$436,603. The energy cost for the system is \$2,054,800.

General Warnings Design Alerts Cost Report Pump Summary Valve Summary Heat Exchanger Summary Reservoir Summary										
Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL	
TOTAL OF ALL MODEL COSTS										2,491,404
Total of All Shown Costs			334,783	101,820	436,603	0	2,054,800	2,054,800	2,491,404	
Items In Sizing			0	0	0	0	0	0	0	
Items Not In Sizing			334,783	101,820	436,603	0	2,054,800	2,054,800	2,491,404	
Pipe Subtotal			280,662	79,853	360,515	0	0	0	360,515	
o	P1	Pipe Pipe	10,795	3,071	13,866	0	0	0	13,866	
o	P2	Pipe Pipe	1,799	512	2,311	0	0	0	2,311	
o	P3	Pipe Pipe	2,699	768	3,466	0	0	0	3,466	
o	P4	Pipe Pipe	2,699	768	3,466	0	0	0	3,466	
o	P5	Pipe Pipe	2,699	768	3,466	0	0	0	3,466	
o	P6	Pipe Pipe	2,699	768	3,466	0	0	0	3,466	
o	P7	Pipe Pipe	1,799	512	2,311	0	0	0	2,311	
o	P8	Pipe Pipe	107,947	30,713	138,660	0	0	0	138,660	
o	P9	Pipe Pipe	1,799	512	2,311	0	0	0	2,311	
o	P10	Pipe Pipe	1,799	512	2,311	0	0	0	2,311	
o	P11	Pipe Pipe	5,397	1,536	6,933	0	0	0	6,933	
o	P12	Pipe Pipe	2,699	768	3,466	0	0	0	3,466	
o	P13	Pipe Pipe	2,699	768	3,466	0	0	0	3,466	
o	P14	Pipe Pipe	5,397	1,536	6,933	0	0	0	6,933	
o	P15	Pipe Pipe	2,699	768	3,466	0	0	0	3,466	
o	P16	Pipe Pipe	2,699	768	3,466	0	0	0	3,466	
o	P17	Pipe Pipe	5,397	1,536	6,933	0	0	0	6,933	
o	P18	Pipe Pipe	2,699	768	3,466	0	0	0	3,466	
o	P19	Pipe Pipe	2,699	768	3,466	0	0	0	3,466	
o	P20	Pipe Pipe	1,799	512	2,311	0	0	0	2,311	
o	P21	Pipe Pipe	1,799	512	2,311	0	0	0	2,311	
o	P22	Pipe Pipe	107,947	30,713	138,660	0	0	0	138,660	
Bend Subtotal			9,884	7,589	17,473	0	0	0	17,473	
o	J3	Bend Bend	2,471	1,897	4,368	0	0	0	4,368	
o	J6	Bend Bend	2,471	1,897	4,368	0	0	0	4,368	
o	J10	Bend Bend	2,471	1,897	4,368	0	0	0	4,368	
o	J19	Bend Bend	2,471	1,897	4,368	0	0	0	4,368	
Pump Subtotal			44,237	14,378	58,616	0	2,054,800	2,054,800	2,113,416	
o	J4	Pump Pump	22,117	7,189	29,306	0	1,027,356	1,027,356	1,056,662	
o	J5	Pump Pump	22,120	7,190	29,310	0	1,027,444	1,027,444	1,056,754	

Figure 14: The Cost Report in the Output window shows the total and individual costs (in U.S. Dollars) for the sized system

With this basis, we will move on to the Initial Cost scenario.

Step 12. Run the Initial Cost scenario and review the results

Load the Initial Cost scenario using the Scenario Manager. This scenario should have all the inputs already specified, so no action is required besides running the model.

Click Run Model on the toolbar or from the Analysis menu. When finished, view the results by clicking the Output button at the bottom of the Solution Progress window.

Go to the Cost Report, as shown in Figure 15. ANS shows all costs in the Cost Report, including those that were not used in the automated sizing. The total cost for this system is \$2,365,110. This includes all

Cooling System - ANS

costs over 10 years. The initial cost, which was the basis for the automated sizing, was the total of the material and installation (\$161,251). This total is listed as the total for the Items in Sizing row, since it was specified to only include the Initial Cost for material and installation on the Sizing Objective section. Individual items that were included in the automated sizing have their costs highlighted in green.

Other costs that are displayed in the Cost Report are Items Not in Sizing. These are items that have costs associated with them, but were not included in the automated sizing.

Note that the Items Not in Sizing total to \$2,203,859. Looking across at the subtotal one can see that all of this cost is in Operation/Energy (i.e., pump power costs). The TOTAL cost is the sum of the Non-Recurring Sub Total and the Recurring Sub Total. For the next scenario this operation/energy cost will be included in the automated sizing.

General Warnings Design Alerts Cost Report Pump Summary Valve Summary Heat Exchanger Summary Reservoir Summary										
Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL	
TOTAL OF ALL MODEL COSTS										2,365,110
Total of All Shown Costs			107,158	54,093	161,251	0	2,203,859	2,203,859	2,365,110	
Items In Sizing			107,158	54,093	161,251	0	0	0	161,251	
Items Not In Sizing			0	0	0	0	2,203,859	2,203,859	2,203,859	
Pipe Subtotal			57,234	36,348	93,582	0	0	0	93,582	
o	P1	Pipe Pipe	2,437	1,494	3,931	0	0	0	3,931	
o	P2	Pipe Pipe	248	195	443	0	0	0	443	
o	P3	Pipe Pipe	372	292	664	0	0	0	664	
o	P4	Pipe Pipe	372	292	664	0	0	0	664	
o	P5	Pipe Pipe	372	292	664	0	0	0	664	
o	P6	Pipe Pipe	372	292	664	0	0	0	664	
o	P7	Pipe Pipe	248	195	443	0	0	0	443	
o	P8	Pipe Pipe	24,372	14,937	39,309	0	0	0	39,309	
o	P9	Pipe Pipe	248	195	443	0	0	0	443	
o	P10	Pipe Pipe	179	152	330	0	0	0	330	
o	P11	Pipe Pipe	536	455	991	0	0	0	991	
o	P12	Pipe Pipe	268	227	496	0	0	0	496	
o	P13	Pipe Pipe	268	227	496	0	0	0	496	
o	P14	Pipe Pipe	536	455	991	0	0	0	991	
o	P15	Pipe Pipe	268	227	496	0	0	0	496	
o	P16	Pipe Pipe	268	227	496	0	0	0	496	
o	P17	Pipe Pipe	536	455	991	0	0	0	991	
o	P18	Pipe Pipe	268	227	496	0	0	0	496	
o	P19	Pipe Pipe	268	227	496	0	0	0	496	
o	P20	Pipe Pipe	179	152	330	0	0	0	330	
o	P21	Pipe Pipe	248	195	443	0	0	0	443	
o	P22	Pipe Pipe	24,372	14,937	39,309	0	0	0	39,309	
Bend Subtotal			736	1,562	2,298	0	0	0	2,298	
o	J3	Bend Bend	217	426	643	0	0	0	643	
o	J6	Bend Bend	217	426	643	0	0	0	643	
o	J10	Bend Bend	151	355	506	0	0	0	506	
o	J19	Bend Bend	151	355	506	0	0	0	506	
Pump Subtotal			49,188	16,183	65,370	0	2,203,859	2,203,859	2,269,229	
o	J4	Pump Pump	24,468	8,045	32,513	0	1,098,129	1,098,129	1,130,642	
o	J5	Pump Pump	24,720	8,137	32,857	0	1,105,729	1,105,729	1,138,587	

Figure 15: The Cost Report in the Output window shows the total and individual costs (in U.S. Dollars) for the sized system

In the Pipes section of the Output window the final pipe sizes from the automated sizing can be seen, along with the Design Requirements status. For this model, the final system used pipe sizes varying from 10 to 16 inches, as seen in Figure 16. The information for the pumps can be seen in the Pump Summary tab in the General Section, which is also shown in Figure 16.

When a monetary cost objective is selected, additional tabs are available that display the Library Sources used for the Cost Report for each of the pipes/junctions. In this case both the Pipe Material and Pipe Installation costs for all of the pipes came from the Steel - ANSI Pipe Costs library which was connected earlier. Information for the Pumps and Bends came from the Cooling System Costs library. If multiple

Cooling System - ANS

cost libraries are used for a category they will all be listed separated by commas. The Library Sources for the Pipes can be seen in Figure 17.

Jct	Results Diagram	Name	Vol. Flow (m3/hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
4	Show	Pump	680.0	188.6	4.226	43.15	70.00	N/A	114.0	N/A	N/A	18.46	10.00
5	Show	Pump	680.0	188.6	4.255	43.45	70.00	N/A	114.7	N/A	N/A	18.31	10.00

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (cm)
1	Pipe	Steel - ANSI	16 inch	STD (schedule 30)	38.73
2	Pipe	Steel - ANSI	12 inch	STD	30.48
3	Pipe	Steel - ANSI	12 inch	STD	30.48
4	Pipe	Steel - ANSI	12 inch	STD	30.48
5	Pipe	Steel - ANSI	12 inch	STD	30.48
6	Pipe	Steel - ANSI	12 inch	STD	30.48
7	Pipe	Steel - ANSI	12 inch	STD	30.48
8	Pipe	Steel - ANSI	16 inch	STD (schedule 30)	38.73
9	Pipe	Steel - ANSI	12 inch	STD	30.48
10	Pipe	Steel - ANSI	10 inch	STD (schedule 40)	25.45
11	Pipe	Steel - ANSI	10 inch	STD (schedule 40)	25.45
12	Pipe	Steel - ANSI	10 inch	STD (schedule 40)	25.45
13	Pipe	Steel - ANSI	10 inch	STD (schedule 40)	25.45
14	Pipe	Steel - ANSI	10 inch	STD (schedule 40)	25.45
15	Pipe	Steel - ANSI	10 inch	STD (schedule 40)	25.45
16	Pipe	Steel - ANSI	10 inch	STD (schedule 40)	25.45
17	Pipe	Steel - ANSI	10 inch	STD (schedule 40)	25.45
18	Pipe	Steel - ANSI	10 inch	STD (schedule 40)	25.45
19	Pipe	Steel - ANSI	10 inch	STD (schedule 40)	25.45
20	Pipe	Steel - ANSI	10 inch	STD (schedule 40)	25.45
21	Pipe	Steel - ANSI	12 inch	STD	30.48
22	Pipe	Steel - ANSI	16 inch	STD (schedule 30)	38.73

Figure 16: Pump and Pipe Sizes after automated sizing is performed

Pipe	Name	Pipe Material	Pipe Installation	Pipe Maintenance
1	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
2	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
3	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
4	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
5	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
6	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
7	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
8	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
9	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
10	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
11	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
12	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
13	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
14	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
15	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
16	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
17	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
18	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
19	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
20	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
21	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	
22	Pipe	Steel - ANSI Pipe Costs	Steel - ANSI Pipe Costs	

Figure 17: Library Sources for the Pipes in the Cost Report

Within the General tab of the Output window at the bottom of the report, the initial cost of the items included in the sizing can be seen, which was \$436,603 for this design, showing that the initial costs changed to \$161,251, a 63% reduction! We can also see that the initial design was feasible, meaning all of the design requirements were met (Figure 18). The ANS module both kept all design requirements satisfied, as well as minimized the initial cost, all in under 3 seconds.

Although there was a 63% reduction in initial costs, the total cost decreased by 5%. Next, we will see how much more the total cost can be reduced by doing a life cycle system sizing.

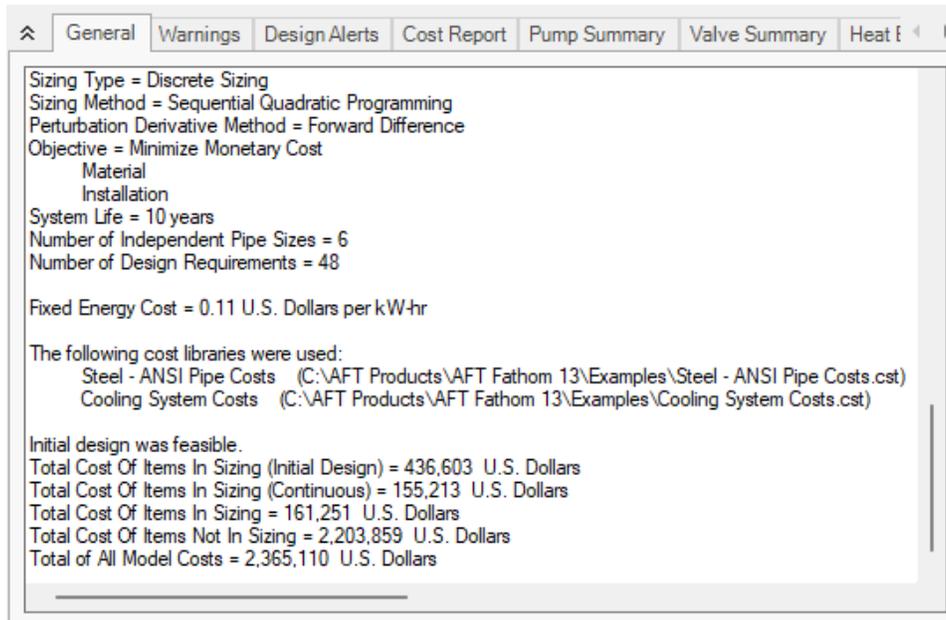


Figure 18: General section of Output window showing initial status of Items in Sizing

Step 13. Size System for Life Cycle Cost over 10 Years

Load the Life Cycle Cost scenario from the Scenario Manager. Go to the Sizing window, and make sure that the Sizing Objective panel is opened. Under Options to Minimize Monetary Cost, choose **Size for Initial and Recurring Costs**. This will move the Energy costs selection to Include in Sizing Objective. The scenario is now complete.

➤ **Click Run from the Analysis menu** to run the automated sizing. When the calculations have finished, click Output to view the results.

Step 14. Compare the Sizing Results

After viewing the Cost Report, it can be seen that the total cost is now \$2,295,100. The Items Not in Sizing row now has zero values, and the Energy columns are highlighted green, reflecting the changes made to include these costs in the objective to minimize monetary cost. The Cost Report is shown in Figure 19.

Taken on its own, this new cost represents a savings of \$196,304 (8%) compared to the Not Sized scenario. Compared to the Initial Cost scenario, which was sized to minimize initial cost only, this new scenario represents a savings of \$70,010. This represents a total 3% cost reduction. In the Initial Cost scenario, the non-recurring cost was \$161,251, while the overall cost was \$2,365,110. Now the non-recurring cost is \$199,356 while the overall cost is \$2,295,100. The initial cost thus increased by \$38,105 in order to reduce the operating cost from \$2,203,859 to \$2,095,743 (a reduction of \$108,116). The source of the operating cost is the cost of power for the pumps. To reduce pump power usage, it makes sense to increase the pipe size and thus reduce frictional losses. The larger pipe sizes can be reviewed

Cooling System - ANS

by looking at the Sizing tab in the Pipe Output section. From the Pump Summary tab it can be seen that the pump power usage decreased slightly from 114 kW to 109 kW at each of the pumps, which is approximately a 5% power requirement decrease at each pump. The Pump Summary and final pipe sizes can be seen in Figure 20.

Considering this cost breakdown between the scenarios highlights the importance of choosing to include the energy costs in the initial cost scenario. If they were not in the Cost Report of the initial cost scenario, the total cost would be \$161,251. Since the life cycle cost scenario total cost is nearly ten times larger, it would appear (at first glance) to be a significant cost increase. However, this would be misleading, since the energy costs still exist in both scenarios, though they are only being minimized in one case. To get a clear comparison between designing for initial cost and life cycle cost one needs to make an apples-to-apples comparison, including all relevant costs in each case, as is done for this example.

By performing both analyses the designer has quantitative data on the impact of initial cost design on operational costs of the system, and can make an informed design choice.

General Warnings Design Alerts Cost Report Pump Summary Valve Summary Heat Exchanger Summary Reservoir Summary									
Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL
TOTAL OF ALL MODEL COSTS									2,295,100
Total of All Shown Costs			135,634	63,723	199,356	0	2,095,743	2,095,743	2,295,100
Items In Sizing			135,634	63,723	199,356	0	2,095,743	2,095,743	2,295,100
Items Not In Sizing			0	0	0	0	0	0	0
Pipe Subtotal			88,162	45,759	133,921	0	0	0	133,921
o P1	Pipe	Pipe	3,637	1,832	5,469	0	0	0	5,469
o P2	Pipe	Pipe	501	277	779	0	0	0	779
o P3	Pipe	Pipe	752	416	1,168	0	0	0	1,168
o P4	Pipe	Pipe	752	416	1,168	0	0	0	1,168
o P5	Pipe	Pipe	752	416	1,168	0	0	0	1,168
o P6	Pipe	Pipe	752	416	1,168	0	0	0	1,168
o P7	Pipe	Pipe	501	277	779	0	0	0	779
o P8	Pipe	Pipe	36,366	18,319	54,685	0	0	0	54,685
o P9	Pipe	Pipe	501	277	779	0	0	0	779
o P10	Pipe	Pipe	501	277	779	0	0	0	779
o P11	Pipe	Pipe	963	660	1,623	0	0	0	1,623
o P12	Pipe	Pipe	482	330	812	0	0	0	812
o P13	Pipe	Pipe	482	330	812	0	0	0	812
o P14	Pipe	Pipe	963	660	1,623	0	0	0	1,623
o P15	Pipe	Pipe	482	330	812	0	0	0	812
o P16	Pipe	Pipe	482	330	812	0	0	0	812
o P17	Pipe	Pipe	963	660	1,623	0	0	0	1,623
o P18	Pipe	Pipe	482	330	812	0	0	0	812
o P19	Pipe	Pipe	482	330	812	0	0	0	812
o P20	Pipe	Pipe	501	277	779	0	0	0	779
o P21	Pipe	Pipe	501	277	779	0	0	0	779
o P22	Pipe	Pipe	36,366	18,319	54,685	0	0	0	54,685
Bend Subtotal			1,874	3,090	4,964	0	0	0	4,964
o J3	Bend	Bend	525	852	1,377	0	0	0	1,377
o J6	Bend	Bend	525	852	1,377	0	0	0	1,377
o J10	Bend	Bend	525	852	1,377	0	0	0	1,377
o J19	Bend	Bend	299	534	833	0	0	0	833
Pump Subtotal			45,597	14,874	60,471	0	2,095,743	2,095,743	2,156,215
o J4	Pump	Pump	22,771	7,427	30,198	0	1,047,049	1,047,049	1,077,248
o J5	Pump	Pump	22,826	7,447	30,273	0	1,048,694	1,048,694	1,078,967

Figure 19: Cost Report for the network after sizing for life cycle cost

Cooling System - ANS

General Warnings Design Alerts Cost Report Pump Summary Valve Summary Heat Exchanger Summary Reservoir Summary													
Jct	Results Diagram	Name	Vol. Flow (m3/hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
4	Show	Pump	680.0	188.6	4.030	41.14	70.00	N/A	108.7	N/A	N/A	18.55	10.00
5	Show	Pump	680.0	188.6	4.036	41.21	70.00	N/A	108.8	N/A	N/A	18.52	10.00

Pipes Sizing Pipe Design Requirements Library Sources					
Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (cm)
1	Pipe	Steel - ANSI	20 inch	STD (schedule 20)	48.90
2	Pipe	Steel - ANSI	18 inch	STD	43.81
3	Pipe	Steel - ANSI	18 inch	STD	43.81
4	Pipe	Steel - ANSI	18 inch	STD	43.81
5	Pipe	Steel - ANSI	18 inch	STD	43.81
6	Pipe	Steel - ANSI	18 inch	STD	43.81
7	Pipe	Steel - ANSI	18 inch	STD	43.81
8	Pipe	Steel - ANSI	20 inch	STD (schedule 20)	48.90
9	Pipe	Steel - ANSI	18 inch	STD	43.81
10	Pipe	Steel - ANSI	18 inch	STD	43.81
11	Pipe	Steel - ANSI	14 inch	STD (schedule 30)	33.66
12	Pipe	Steel - ANSI	14 inch	STD (schedule 30)	33.66
13	Pipe	Steel - ANSI	14 inch	STD (schedule 30)	33.66
14	Pipe	Steel - ANSI	14 inch	STD (schedule 30)	33.66
15	Pipe	Steel - ANSI	14 inch	STD (schedule 30)	33.66
16	Pipe	Steel - ANSI	14 inch	STD (schedule 30)	33.66
17	Pipe	Steel - ANSI	14 inch	STD (schedule 30)	33.66
18	Pipe	Steel - ANSI	14 inch	STD (schedule 30)	33.66
19	Pipe	Steel - ANSI	14 inch	STD (schedule 30)	33.66
20	Pipe	Steel - ANSI	18 inch	STD	43.81
21	Pipe	Steel - ANSI	18 inch	STD	43.81
22	Pipe	Steel - ANSI	20 inch	STD (schedule 20)	48.90

Figure 20: Pump and Pipe Sizes after automated sizing is performed for life cycle cost

Step 15. Size System for Life Cycle Cost with Initial Cost Limit

While the energy cost savings from life cycle cost sizing are desirable, the budget for the project may not support the higher initial cost that is required. In that case, we can apply an initial cost limit to the life cycle cost sizing.

Load the Initial Cost Limit scenario from the Scenario Manager.

➤ **Return to the Sizing Objective panel** and select the option for Size for Initial and Recurring Costs and check the box next to Initial Cost Limit. We will set the limit as \$175,000, which is slightly more than the initial cost of the scenario that minimized initial cost at the expense of an increased life cycle cost. Run the model and go to the Output tab.

A summary of the results with the initial cost limit compared to each of the initial and life cycle cost sizing scenarios for a 10 year life cycle can be seen in Table 1. Notice that while the material and installation costs were limited to \$175,000, the total costs over a 10 year lifetime are decreased by about \$48,392 compared to the Initial Cost scenario. The initial cost increased from \$161,251 to \$175,158: a \$13,907 increase. Therefore, a \$13,907 initial cost increase leads to a life cycle decrease of \$48,392. This savings comes from the reduction pump energy costs. These reduced energy costs were achieved by increasing the Main pipes (P1 & P22), pipe P8, and the Header pipes while decreasing the pipes in Cooler Piping.

Figure 21 shows a comparison of the pipe sizes throughout the system for the Initial Cost scenario, the Life Cycle Cost scenario, and the Life Cycle Cost with Initial Cost Limit scenario. Note that this diagram was generated outside of AFT Fathom, but sized results can be shown for each scenario individually using Workspace Layers.

Table 1: Cost Summary of Sizing Runs for Cooling System

Scenario	Material	Installation	Non-Recurring Sub Total	Operation/Energy	TOTAL	Reduction
Not Sized	334,783	101,820	436,603	2,054,800	2,491,404	-
Initial Cost	107,158	54,093	161,251	2,203,859	2,365,110	126,294
Life Cycle Cost	135,634	63,723	199,356	2,095,743	2,295,100	196,304
Initial Cost Limit	117,815	57,343	175,158	2,141,559	2,316,718	174,686

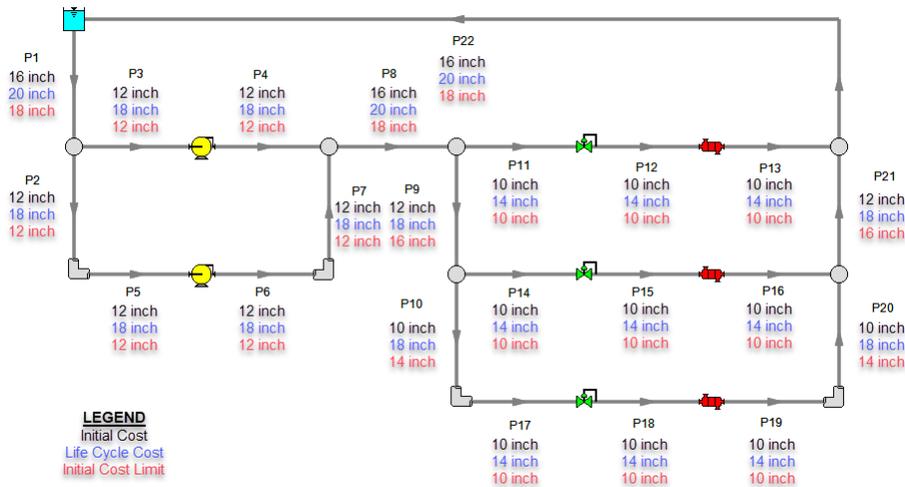


Figure 21: Pipe sizes selected by the ANS module for the life cycle cost and life cycle cost with an initial cost limit compared to sizing for initial cost only

Step 16. Automated Sizing with Pump Curve Data

As discussed in the section *Summarizing the pump selection process*, once the pump is sized then actual pumps can be modeled. The actual pump should closely match the sizing results in the following

areas: generated head at the design flow, efficiency at the design flow, and cost. We will apply a pump curve for the case that was sized for life cycle cost to complete the pump selection process.

Reviewing the results for the life cycle cost scenario one can see that the sized system calls for a pump of about 109 kW that generates about 41 meters of head at 680 m³/hr. The nominal efficiency used in the sizing part of the analysis was 70%. The material cost for such a pump was about \$22,800 and the installation cost was about \$7,400. Note that if no actual pump can be found that reflects these requirements, then the phase 1 of the pump selection process described above should be repeated with better performance and/or cost data for the pump.

Based on the sizing analysis, we will choose a pump from the (fictitious) PowerFlow Pump Company. Its model JW120 is a close match for the requirements of a 10-year life cycle design. The model JW120 has the following characteristics: at 680 m³/hr its head is 45 meters and efficiency is 70%. Its material cost is \$25,000 and installation cost is \$8,000. To set up a new scenario for the system with the candidate pump data:

1. Load the Actual Pump Curve scenario and go to the Output window. Currently it should be identical to the parent scenario Life Cycle Cost. Run the scenario.
2. Open the Edit menu and select **Transfer Results to Initial Guesses**. When the Transfer Results Options window appears, select **Calculated Pipe Sizes Only** and hit **OK**.

Note: The pipe sizes in Model Data are now equal to the pipe sizes selected by ANS for the Life Cycle Cost scenario. Starting from a more optimal solution can aid convergence and in this case will improve solver execution time.

3. Go back to the Workspace. In the properties window for each of the **pumps** select **PowerFlow Pump JW120** from the Library Jct list to import the pump data from the library as shown in Figure 22, then click OK.
4. In the Control Valve properties window for **J13**, change the Valve Type to **Flow Control (FCV)**, and enter a Flow Setpoint of **450 m³/hr**. Uncheck Always Control (Never Fail), Now that we have changed the control valve model, we will need to add a new Design Requirement in order to account for the minimum pressure drop.
5. Go to the **Sizing** window, navigate to the **Design Requirements** panel, select the **Control Valves** button at the top of the window, and **apply** the Design DP design requirement to J13.

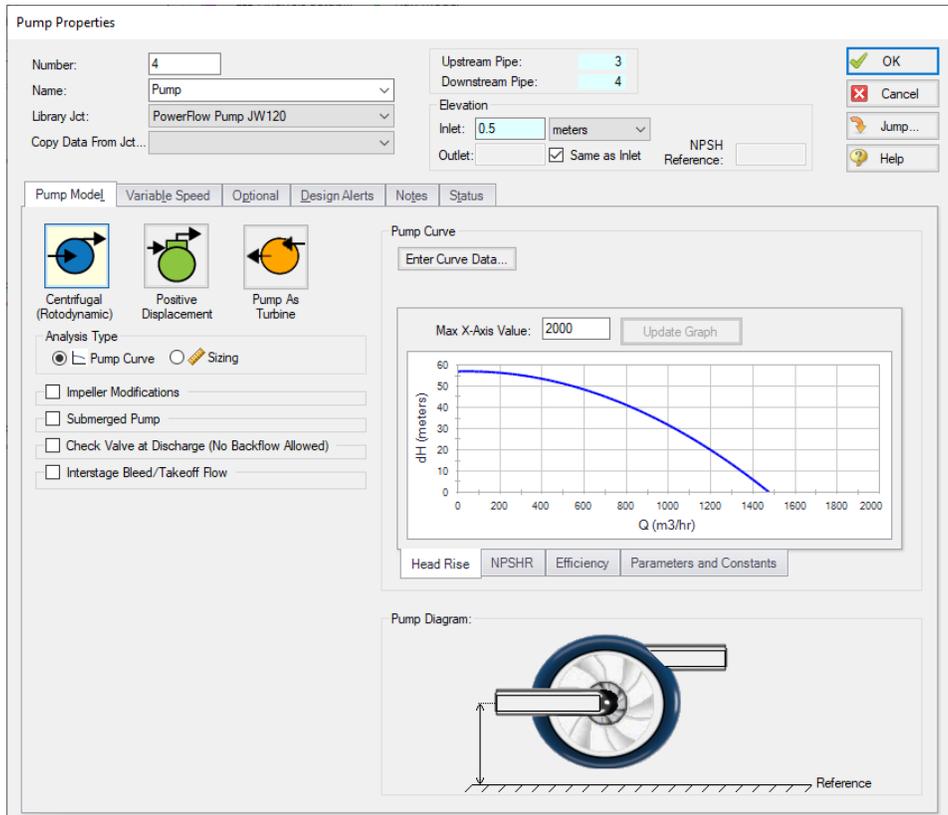


Figure 22: Pump Properties window with the PowerFlow Pump JW120 selected

The necessary changes are now complete to size the system with the updated pump and flow control valve data.

➤ **Select Run from the Analysis menu** to perform the automated sizing - this may take some time.

When the solver has finished, click Output to review the results. The overall cost for the system is about \$2,476,563. Clicking the Pump Summary tab shows that the operating point with the actual pump curve is similar to the sizing scenario.

Now we have sized the system for use with an actual pump.

Cooling System - ANS

General Warnings Design Alerts Cost Report Pump Summary Valve Summary Heat Exchanger Summary Reservoir Summary										
Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL	
TOTAL OF ALL MODEL COSTS										2,476,563
Total of All Shown Costs			107,971	53,910	161,881	0	2,314,682	2,314,682	2,476,563	
Items In Sizing			107,971	53,910	161,881	0	2,314,682	2,314,682	2,476,563	
Items Not In Sizing			0	0	0	0	0	0	0	
Pipe Subtotal			57,234	36,348	93,582	0	0	0	93,582	
o	P1	Pipe	Pipe	2,437	1,494	3,931	0	0	0	3,931
o	P2	Pipe	Pipe	248	195	443	0	0	0	443
o	P3	Pipe	Pipe	372	292	664	0	0	0	664
o	P4	Pipe	Pipe	372	292	664	0	0	0	664
o	P5	Pipe	Pipe	372	292	664	0	0	0	664
o	P6	Pipe	Pipe	372	292	664	0	0	0	664
o	P7	Pipe	Pipe	248	195	443	0	0	0	443
o	P8	Pipe	Pipe	24,372	14,937	39,309	0	0	0	39,309
o	P9	Pipe	Pipe	248	195	443	0	0	0	443
o	P10	Pipe	Pipe	179	152	330	0	0	0	330
o	P11	Pipe	Pipe	536	455	991	0	0	0	991
o	P12	Pipe	Pipe	268	227	496	0	0	0	496
o	P13	Pipe	Pipe	268	227	496	0	0	0	496
o	P14	Pipe	Pipe	536	455	991	0	0	0	991
o	P15	Pipe	Pipe	268	227	496	0	0	0	496
o	P16	Pipe	Pipe	268	227	496	0	0	0	496
o	P17	Pipe	Pipe	536	455	991	0	0	0	991
o	P18	Pipe	Pipe	268	227	496	0	0	0	496
o	P19	Pipe	Pipe	268	227	496	0	0	0	496
o	P20	Pipe	Pipe	179	152	330	0	0	0	330
o	P21	Pipe	Pipe	248	195	443	0	0	0	443
o	P22	Pipe	Pipe	24,372	14,937	39,309	0	0	0	39,309
Bend Subtotal			736	1,562	2,298	0	0	0	2,298	
o	J3	Bend	Bend	217	426	643	0	0	0	643
o	J6	Bend	Bend	217	426	643	0	0	0	643
o	J10	Bend	Bend	151	355	506	0	0	0	506
o	J19	Bend	Bend	151	355	506	0	0	0	506
Pump Subtotal			50,000	16,000	66,000	0	2,314,682	2,314,682	2,380,682	
o	J4	Pump	Pump	25,000	8,000	33,000	0	1,159,983	1,159,983	1,192,983
o	J5	Pump	Pump	25,000	8,000	33,000	0	1,154,699	1,154,699	1,187,699

Figure 23: Cost Report for system with actual Pump and FCV data sized for 10 year life cycle cost

Cooling System, Multiple Design Cases - ANS

Summary

Some pipe system designs have multiple operating conditions to satisfy. For instance, the cooling water example discussed in the previous example may have a requirement that it operate with one of the pumps turned off. This can be modeled in the ANS module by taking advantage of its ability to model multiple systems concurrently.

Note: This example can only be run if you have a license for the ANS module.

Topics Covered

- Defining dependent Design Cases
- Using Duplicate Special to create Dependent Designs
- Specifying fractional pump costs with Dependent Design Cases
- Using multiple Dependent Design Cases

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the [Beginner - Three-Reservoir - ANS](#) example, and is familiar with the basics of ANS analysis.

Since this example is a continuation of the [Cooling System - ANS](#) example, it is recommended to complete that example first before beginning this example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Cooling System - ANS.fth***
- ***Metric - Cooling System Multiple Design Cases - ANS.fth***
- ***Cooling System.dat*** - engineering library
- ***Cooling System Costs.cst*** - cost library for ***Cooling System.dat***
- ***Steel - ANSI Pipe Costs.cst*** - cost library for ***Steel - ANSI*** pipes

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **Metric - Cooling System - ANS.fth** example file and save it as a new file. Delete the annotations and hide the layers for Section A through E. The Workspace should look like Figure 1 below.

Browse to the Life Cycle Cost scenario in Scenario Manager. Create a child scenario from this called *Dependent Design Cases* to work in for this example.

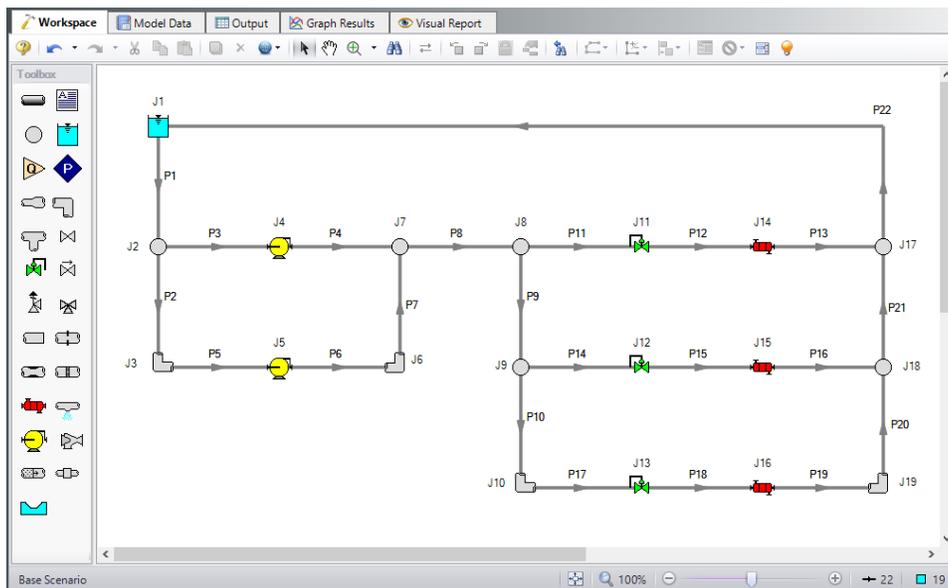


Figure 1: Workspace for Cooling System model

Step 3. Enable Dependent Design

When modeling multiple design cases, the different cases need to be broken up into a primary design case, of which there is one, and dependent design cases, of which there can be any number.

The primary design case can be any of the design cases that are being addressed in your model. Typically, it will represent the primary operating condition of the pipe system. The dependent design cases represent the same physical operating system, but are used to represent differing operating conditions. If it is not clear which case should be the primary case, just pick any of the cases and call that the primary one. Design cases other than the primary case are referred to as dependent design cases. The reason

why we choose a primary case and refer to the other cases as dependent cases will become clearer as we progress through this topic.

For this example the primary design case is the system running with all pumps turned on. We want to also consider the case where only one pump is running. The one pump running case will be created as a dependent design case.

Dependent design case modeling is enabled in the Sizing Objective panel.

➤ **In the Dependent Design Case scenario go to the Sizing window and to the Sizing Objective panel.** Check the box next to **Enable Dependent Design Cases**. A new button will now be available for the Dependent Design Cases panel in the Sizing Navigation Panel as shown in Figure 2.

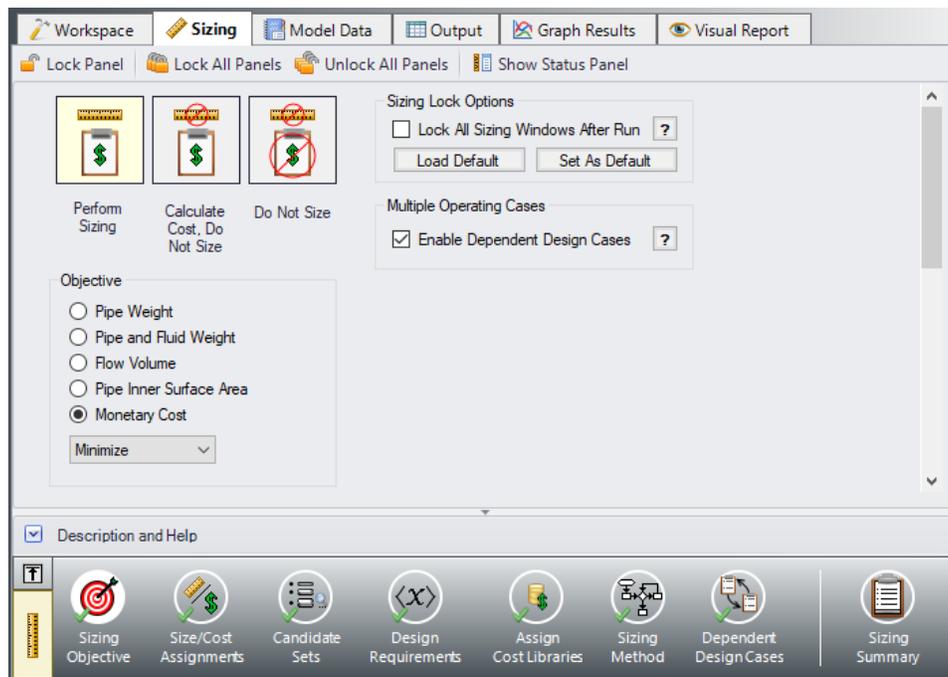


Figure 2: The Dependent Design Cases panel is enabled from the Sizing Objective panel

Step 4. Setting up the Dependent Design Case

A. Using Duplicate Special

The easiest way to set up a dependent design case once it has been enabled is to use Duplicate Special, which will allow us to make an exact duplicate of the model in order to setup the additional operating cases. Note that you should fully define the model and all the sizing settings for the primary case before creating the dependent cases, as has been done in this model.

To create the dependent cases:

1. Make sure that the scenario where the dependent cases will be created is loaded. In this case, you should have the Dependent Design Cases scenario loaded.
2. Go to the **Workspace**.
3. Go to the **Edit menu** and choose **Select All**.
4. Go to the **Edit menu** again and choose **Duplicate Special** (see Figure 3).
5. Choose an increment for the pipe and junction numbers. The number should be large enough to avoid conflicts with numbers used for other pipes and junctions in this and all other scenarios. In this case use **100**.
6. Select the **Make Dependent Design Case** check box.
7. Click **OK** and move the duplicated version of the primary design case to an open area in the Workspace. The model should now appear similar to Figure 4.

Note that Duplicate Special also allows you to do the following:

- Create a group for the dependent design case. This can simplify managing the pipe and junction objects in the design case.
- Hide the pipe and junction in the output. You may want to do this for reasons to be discussed later in this topic.
- Copy Layer Style Settings From Original Objects. If the visibility of the Section A through E layers is toggled back on, they will apply the same colors to both the duplicate and original model if this option was checked while creating the duplicate.
- Keep Common Size Groups. This feature is useful when using Duplicate Special for reasons other than dependent case creation, and can't be used when making a dependent design case.
- Clear the Design Requirements. For reasons to be discussed, frequently dependent design cases will employ different design requirements than the primary design case. This option allows you to clear all the current requirements, so new ones can be assigned.

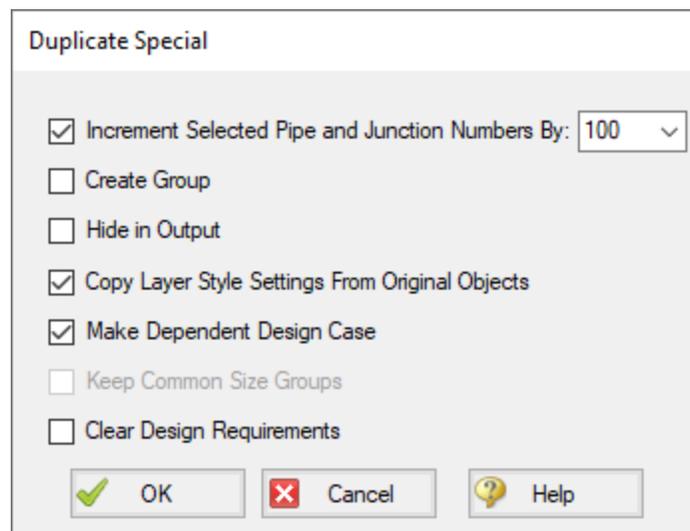


Figure 3: Duplicate Special simplifies creation of dependent design cases

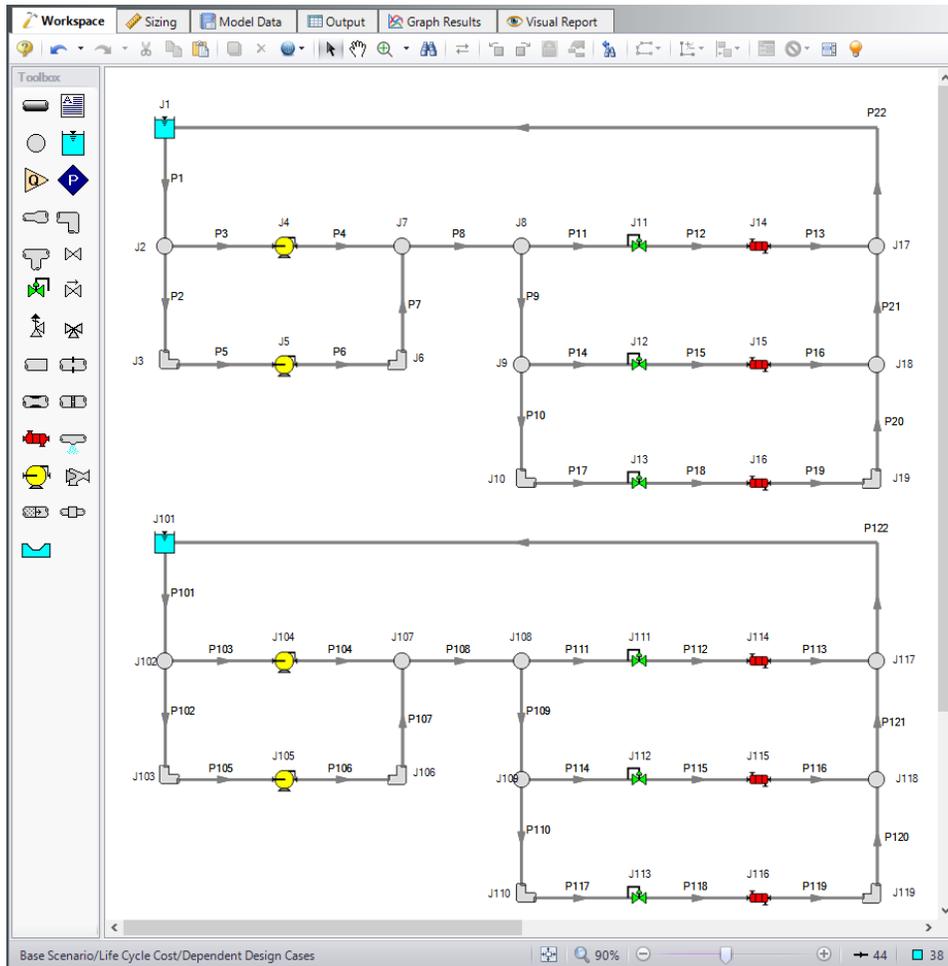


Figure 4: Workspace with the Dependent Design Case created for the Cooling System model

B. Apply the Dependent Design Operating Conditions

The dependent design case capability takes advantage of the ANS module's ability to run multiple models in the same Workspace. The pipes and junctions in Figure 4 that are numbered greater than 100 (i.e., the dependent design case) represent the same pipes and junctions as those numbered less than 100 (i.e., the primary design case). It is therefore critically important that only certain data in the dependent case is changed.

Frequently the Design Requirements or Special Conditions on the pipes or junctions will be changed. However, data such as a pipe's length or pump's performance should never change. The reason is that once this kind of data is changed, the pipe or junction is no longer the same pipe or junction, and thus the purpose of the dependent link is invalidated. The ANS module does not prevent changing data that should not be changed, but when the model is run it will check to make sure all data that should be the same actually is the same.

In this example we are analyzing the case where one pump in the system is shut off, so we will use Special Conditions on the pump. With one pump shutoff we will not have enough flow to supply all three heat exchangers, so we will also close one of the heat exchangers as well as a control valve.

➤ **Change the Special Conditions of J105 to Pump Off No Flow.**

➤ **Change the Special Conditions of J111 to Closed.**

Selecting these junctions on the Workspace and using the F4 keyboard shortcut is an option for applying these special conditions quickly. The dependent case should now appear as shown in Figure 5.

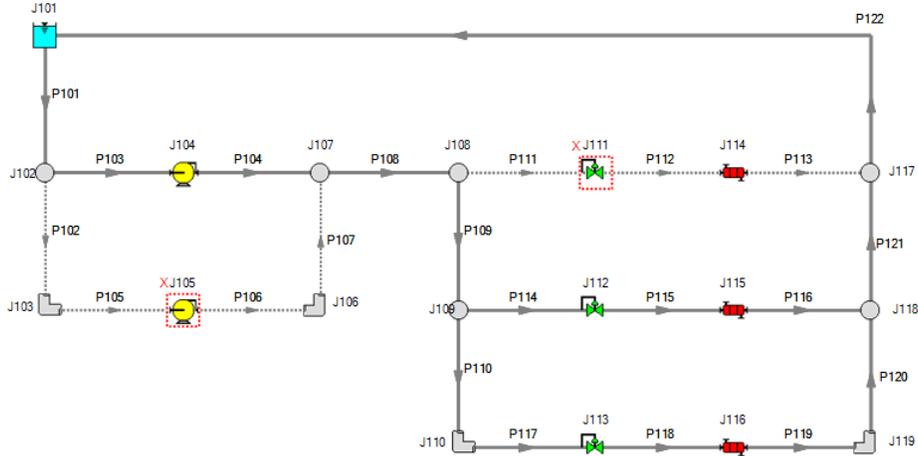


Figure 5: Flow off for Pump 105 and Control Valve 111 in the Dependent Design Case

We will now need to adjust the Design Requirements for the model. Go to the Design Requirements panel on the Sizing window.

Currently it can be seen that Duplicate Special maintained all of the same design requirement applications from the primary case in the dependent case. While most of the design requirements used in the primary case still apply, we will need to remove the Design Requirements from the closed sections of the model. If the model were run without making any changes, the ANS module would not be able to find a feasible system, since it is impossible to achieve a minimum flow of 450 m³/hr in Heat Exchanger 114 now that it has been forced closed. A similar problem exists with the minimum NPSH margin being applied to Pump 105, which is also closed.

The flow requirements for Heat Exchanger 115 and 116 will also need to be reconsidered. Since there is only one pump running a lower supply flow rate is expected, so the minimum flow rate required by the coolers should be lowered to 340 m³/hr.

The pressure drop across the control valves still needs to be a minimum of 0.345 bar. For J112, this is accomplished by using a Design Requirement whereas J113 is defined as a constant pressure drop control valve (PDCV) with the required pressure drop.

Apply the changes as follows:

1. Click the Pumps button to go to the pump Design Requirements
2. Uncheck the box next to J105 to remove the NPSH requirement from Pump 105 in the dependent case (Figure 6).

3. Click the Control Valves button to go to the control valve Design Requirements.
4. Uncheck the box next to J111 to remove the pressure drop requirement from the closed control valve (Figure 7).
5. Return to the Workspace and open the Control Valve Properties window for J112.
6. Change the Flow Setpoint to 340 m³/hr(Figure 8).

No other changes are needed, so the Dependent Design Case scenario is now complete.

➤ **Select Run from the Analysis menu** to perform the automated sizing, and go to the Output window once the run is complete to view the results.

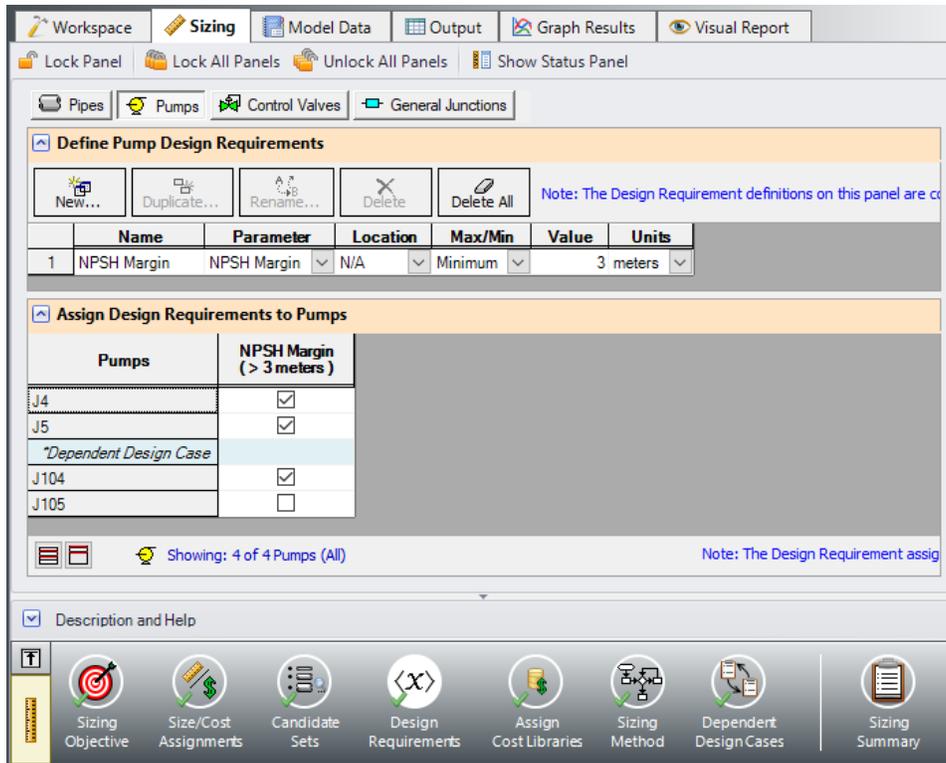


Figure 6: Pump Design Requirements for the dependent design case

The screenshot shows a software interface with a top menu bar containing 'Workspace', 'Sizing', 'Model Data', 'Output', 'Graph Results', and 'Visual Report'. Below this is a toolbar with 'Lock Panel', 'Lock All Panels', 'Unlock All Panels', and 'Show Status Panel'. The main area is divided into two sections:

Define Control Valve Design Requirements

This section includes buttons for 'New...', 'Duplicate...', 'Rename...', 'Delete', and 'Delete All'. A table below lists design requirements:

	Name	Parameter	Location	Max/Min	Value	Units
1	Design DP	Pressure Drop Static	N/A	Minimum	0.345 bar	

Assign Design Requirements to Control Valves

This section shows a table for assigning requirements to control valves:

Control Valves	Design DP (> 0.345 bar)
J11	<input checked="" type="checkbox"/>
J12	<input checked="" type="checkbox"/>
J13	<input type="checkbox"/>
<i>*Dependent Design Case</i>	
J111	<input type="checkbox"/>
J112	<input checked="" type="checkbox"/>
J113	<input type="checkbox"/>

At the bottom of the interface, there is a 'Description and Help' section and a navigation bar with icons for 'Sizing Objective', 'Size/Cost Assignments', 'Candidate Sets', 'Design Requirements', 'Assign Cost Libraries', 'Sizing Method', 'Dependent Design Cases', and 'Sizing Summary'.

Figure 7: Control Valve Design Requirements for the dependent design case

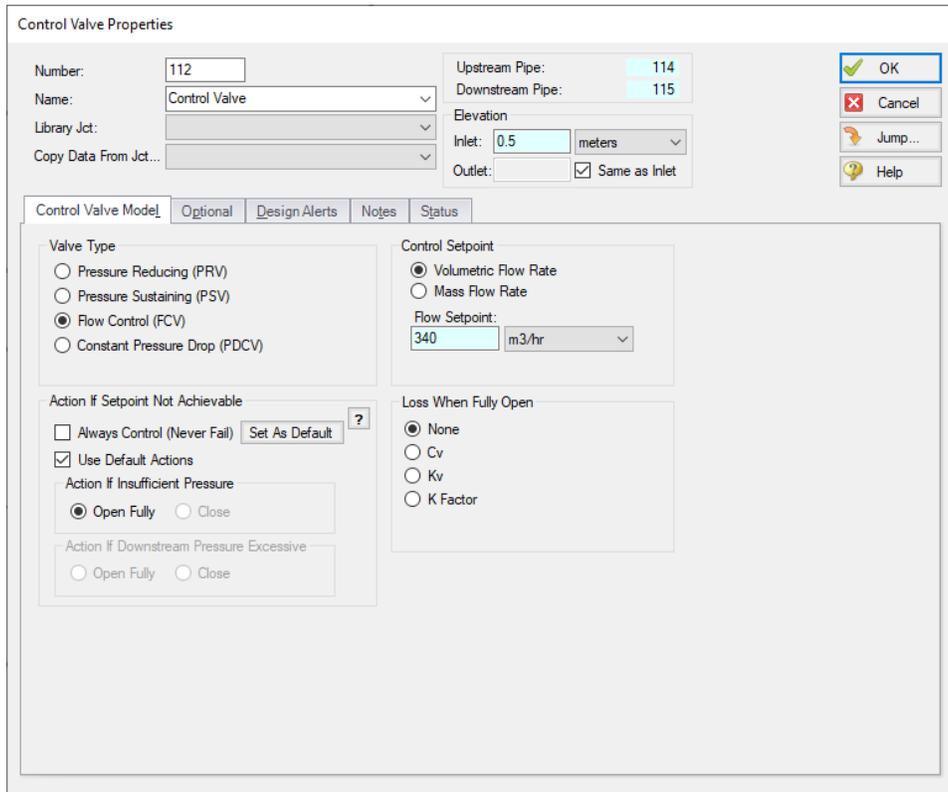


Figure 8: Flow Setpoint adjusted in the dependent design case for FCV J112

Summary

The cost report for the Dependent Design Case scenario can be seen in Figure 9 below.

Note that the pipes and junctions in the dependent case are separate entities for modeling purposes, but represent the same pipes and junctions as the primary case. Thus when performing automated sizing, we only count the cost of the pipes and junctions once. This occurs in the primary design case, which in our model is the two pumps running case. This is why only the primary design case pipes are shown in the cost report.

With one exception, it is not possible for a model that includes a dependent design case to yield a better design than the primary case alone. At best, it can yield the same design, such as in this scenario. This is due to the fact that the dependent design case imposes additional restrictions on the solution by adding additional design requirements. If the design for the primary design case is not sufficient for the dependent design case, then the design must be changed, which will increase cost in some manner.

Turning off Pump 105 in this scenario required no adjustments in the pipe sizes, so the initial sizing results remained unchanged.

Cooling System, Multiple Design Cases - ANS

General Warnings Design Alerts Cost Report Pump Summary Valve Summary Heat Exchanger Summary Reservoir Summary										
Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL	
TOTAL OF ALL MODEL COSTS										2,295,100
Total of All Shown Costs			135,634	63,723	199,356	0	2,095,743	2,095,743	2,295,100	
Items In Sizing			135,634	63,723	199,356	0	2,095,743	2,095,743	2,295,100	
Items Not In Sizing			0	0	0	0	0	0	0	
Pipe Subtotal			88,162	45,769	133,921	0	0	0	133,921	
o P1	Pipe	Pipe	3,637	1,832	5,469	0	0	0	5,469	
o P2	Pipe	Pipe	501	277	779	0	0	0	779	
o P3	Pipe	Pipe	752	416	1,168	0	0	0	1,168	
o P4	Pipe	Pipe	752	416	1,168	0	0	0	1,168	
o P5	Pipe	Pipe	752	416	1,168	0	0	0	1,168	
o P6	Pipe	Pipe	752	416	1,168	0	0	0	1,168	
o P7	Pipe	Pipe	501	277	779	0	0	0	779	
o P8	Pipe	Pipe	36,366	18,319	54,685	0	0	0	54,685	
o P9	Pipe	Pipe	501	277	779	0	0	0	779	
o P10	Pipe	Pipe	501	277	779	0	0	0	779	
o P11	Pipe	Pipe	963	660	1,623	0	0	0	1,623	
o P12	Pipe	Pipe	482	330	812	0	0	0	812	
o P13	Pipe	Pipe	482	330	812	0	0	0	812	
o P14	Pipe	Pipe	963	660	1,623	0	0	0	1,623	
o P15	Pipe	Pipe	482	330	812	0	0	0	812	
o P16	Pipe	Pipe	482	330	812	0	0	0	812	
o P17	Pipe	Pipe	963	660	1,623	0	0	0	1,623	
o P18	Pipe	Pipe	482	330	812	0	0	0	812	
o P19	Pipe	Pipe	482	330	812	0	0	0	812	
o P20	Pipe	Pipe	501	277	779	0	0	0	779	
o P21	Pipe	Pipe	501	277	779	0	0	0	779	
o P22	Pipe	Pipe	36,366	18,319	54,685	0	0	0	54,685	
Bend Subtotal			1,874	3,090	4,964	0	0	0	4,964	
o J3	Bend	Bend	525	852	1,377	0	0	0	1,377	
o J6	Bend	Bend	525	852	1,377	0	0	0	1,377	
o J10	Bend	Bend	525	852	1,377	0	0	0	1,377	
o J19	Bend	Bend	299	534	833	0	0	0	833	
Pump Subtotal			45,597	14,874	60,471	0	2,095,743	2,095,743	2,156,215	
o J4	Pump	Pump	22,771	7,427	30,198	0	1,047,049	1,047,049	1,077,248	
o J5	Pump	Pump	22,826	7,447	30,273	0	1,048,694	1,048,694	1,078,967	

Figure 9: Cost Report for the Dependent Design Case scenario

Hot Water System - ANS

Summary

This example will take an existing hot water system design, scale it up, and size the pipes to meet a set of system requirements at a minimal cost. The system cost is included to demonstrate the need for sizing. The system must meet the following requirements:

- Minimum 0.7 bar (70 kPa) drop across flow control valves in all cases
- Maximum 2.5 meters/sec velocity in all pipes in all cases
- No NPSH violations
- All pumps must operate between 80% to 100% of the BEP (Best Efficiency Point)
- System must also function with Pump A turned off with no BEP restrictions
- The flow rate through the heat exchangers is controlled by FCVs set to 680 m³/hr.

Note: This example can only be run if you have a license for the ANS module.

Topics Covered

- Defining pump Design Requirements
- Sizing the system to minimize initial costs
- Using Dependent Design Cases

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the [Beginner - Three-Reservoir - ANS](#) example, and is familiar with the basics of ANS analysis.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Hot Water System.fth***
- ***Metric - Hot Water System - ANS.fth***

- **Hot Water System.dat** - engineering library
- **Hot Water System Costs.cst** - cost library for **Hot Water System.dat**
- **Steel - ANSI Pipe Costs.cst** - cost library for **Steel - ANSI** pipes

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **Metric - Hot Water System.fth** example file and save it as a new file. We will not need the existing child scenarios for this example: right click on the Base Scenario in the Scenario Manager and select **Delete All Children**.

Although the model is fully defined, a few alterations are necessary for this example to scale up the system. Open the Workspace Layers tab in the Quick Access Panel and load the Layer Preset **ANS Sizing Group View** to display the sizing group colors. The Workspace should now appear as shown in Figure 1 below. Colors for each sizing group can be toggled individually by toggling their respective layer.

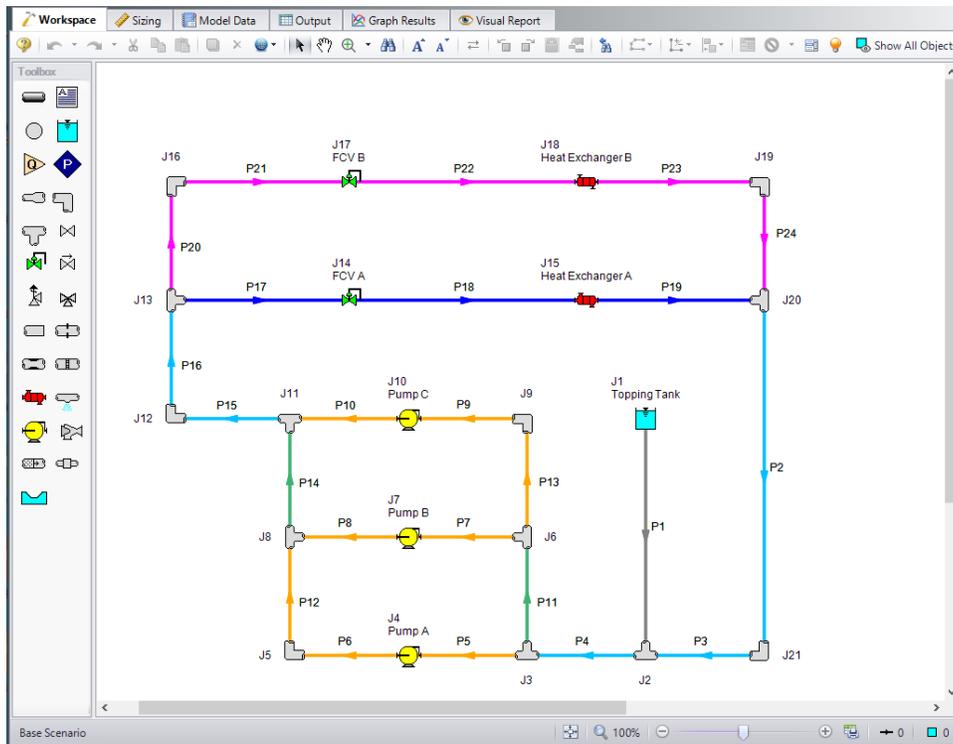


Figure 1: Workspace for the hot water system model with Common Size Groups indicated by color

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate ANS and select Network to enable the ANS module for use. A new group will appear in Analysis Setup titled Automatic Sizing. Click OK to save the changes and exit Analysis Setup. A new Primary Window tab will appear between Workspace and Model Data titled Sizing. Open the Analysis menu to see the new option called Automatic Sizing. From here you can quickly toggle between Not Used mode (normal AFT Fathom) and Network (ANS mode).

Step 4. Define the Pipes and Junctions Group

For this example, the existing system is being scaled up. As an initial design, set all pipe diameters to 20 inches and increase the flow setpoints of the control valves to 680 m³/hr.

Make the following changes to these pipes junctions

1. All Pipes
 - a. Size = 20 inch
 - b. Type = STD (schedule 20)
2. J14 & J17 Control Valves
 - a. Flow Setpoint = 680 m³/hr

Step 5. Configure Sizing Settings

We will need to define the sizing settings to account for the three given Design Requirements while minimizing the initial costs for the system. To do this, we will follow the Sizing steps as outlined below.

A. Sizing Objective

The Sizing Objective panel should be selected by default in the Sizing Navigation panel.

Select Perform Sizing.

For the Objective, make sure that **Monetary Cost** is selected, and **Minimize** is chosen from the drop-down.

We will also need to specify which types of monetary cost we want to size for. Choose **Size for Initial and Recurring Costs**, if not selected already, to move the Material and Installation costs to the column to be sized.

Set the System Life to 10 years.

Under Energy Cost, select Use This Energy Cost Information and enter 0.12 U.S. Dollars Per kW-hr.

B. Size/Cost Assignments

On the toolbar at the bottom of the window select the Size/Cost Assignments button. The Size/Cost Assignments window allows the user to define what will be sized in the model.

To increase the analysis speed and reduce complexity, it is useful to create Common Size Groups to group the pipes in different sections of the model. For this system we have defined 5 groups to be sized, as can be seen in Figure 2. We do not want to size pipe P1, but we will include it in the cost calculation for the sake of comparison later.

➤ **Create the Common Size Groups** shown in Figure 1 and Figure 2 in the model by either using the radio buttons in the table, or by selecting the pipes in the Workspace and adding them to groups from the right click menu.

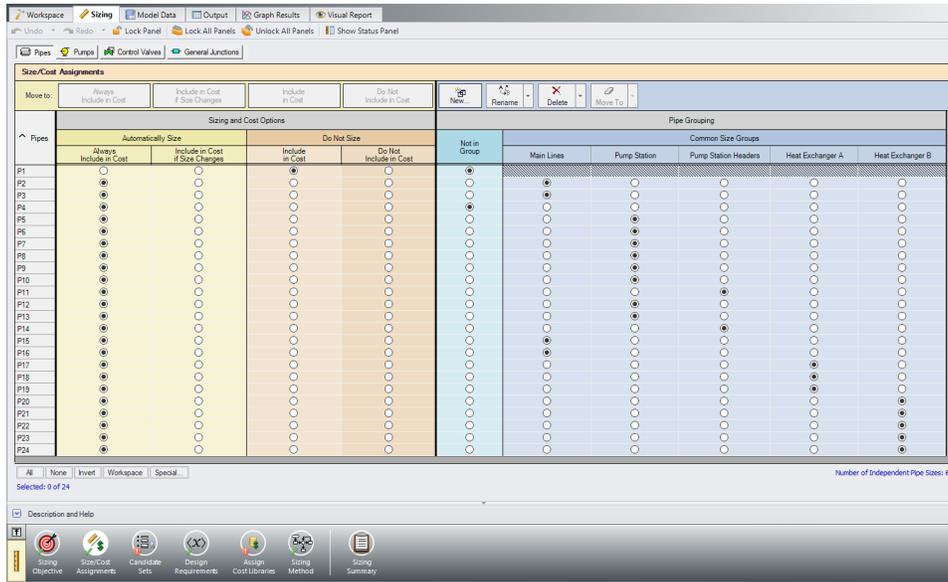


Figure 2: Size/Cost Assignments using Common Size Groups to simplify calculations

Besides the pipes, it is also desired to include the pumps in the sizing.

➤ **Select the Pumps** button then set each of the pumps to be included in the Cost Report and Sizing.

C. Candidate Sets

Click on the Candidate Sets button to open the Candidate Sets panel.

Click New under Define Candidate Sets and name it *STD Steel 1"-36"*. In the Select Pipe Sizes window, select Steel - ANSI from the dropdown menu, expand the STD group (make sure the Sort selection on the bottom is Type, Schedule, Class), then add all pipes from 1 inch to 36 inch to the Candidate Set list on the right. Click OK to close the Select Pipe Sizes window.

In the bottom section of the window, check that each of the Common Size Groups have been assigned to use the existing Candidate Set.

D. Design Requirements

Select the Design Requirements button.

For this model we only have three given Design Requirements which will need to be applied for the pipes, control valves, and pumps. We will start with the pipes:

1. Ensure the Pipes button at the top of the window is selected.
2. Click the New button under Define Pipe Design Requirements.
3. Enter a name when prompted: *Max Pipe Velocity*.
4. In the table, select Velocity as the Parameter.
5. Choose Maximum for Max/Min, and enter 2.5 meters/sec.

Now that the requirement is defined, make sure that it has been applied to all pipes in the model by checking the box next to each of the Common Size Groups in the bottom section of the window, as shown in Figure 3.

Repeat the same process as above by clicking the Pumps, then Control Valves button to apply the Design Requirements for the Pumps and Control Valves as follows, and as is shown in Figure 5 and Figure 4 respectively.

- All of the Pumps must have a minimum % Flow of BEP of 80% and a maximum % Flow of BEP of 100%
- Both Control Valves must have a minimum stagnation pressure drop of 0.7 bar (70 kPa)

Note: The Design Requirements are not enforced when the model is not being sized. Instead, the design requirements are treated as Design Alerts, which is useful to assess the feasibility of the initial design.

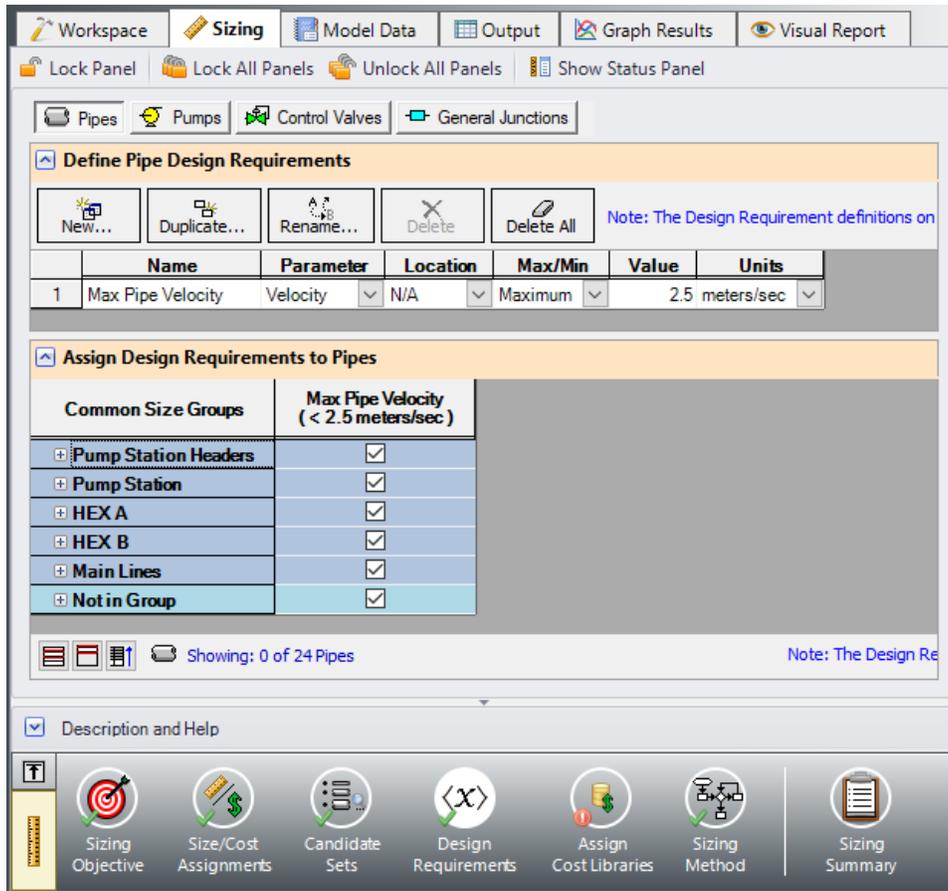


Figure 3: Pipe Design Requirements

Workspace | Sizing | Model Data | Output | Graph Results | Visual Report

Lock Panel | Lock All Panels | Unlock All Panels | Show Status Panel

Pipes | Pumps | Control Valves | General Junctions

Define Pump Design Requirements

New... Duplicate... Rename... Delete Delete All Note: The Design Requirement definitions o

	Name	Parameter	Location	Max/Min	Value	Units
1	Min % BEP	% Flow of BEP	N/A	Minimum	80 Percent	
2	Max % BEP	% Flow of BEP	N/A	Maximum	100 Percent	

Assign Design Requirements to Pumps

Pumps	Min % BEP (> 80 Percent)	Max % BEP (< 100 Percent)
J4 (Pump C)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
J7 (Pump B)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
J10 (Pump A)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

Showing: 3 of 3 Pumps (All) Note: The Design R

Description and Help

Sizing Objective
 Size/Cost Assignments
 Candidate Sets
 Design Requirements
 Assign Cost Libraries
 Sizing Method
 Sizing Summary

Figure 4: Pump Design Requirements

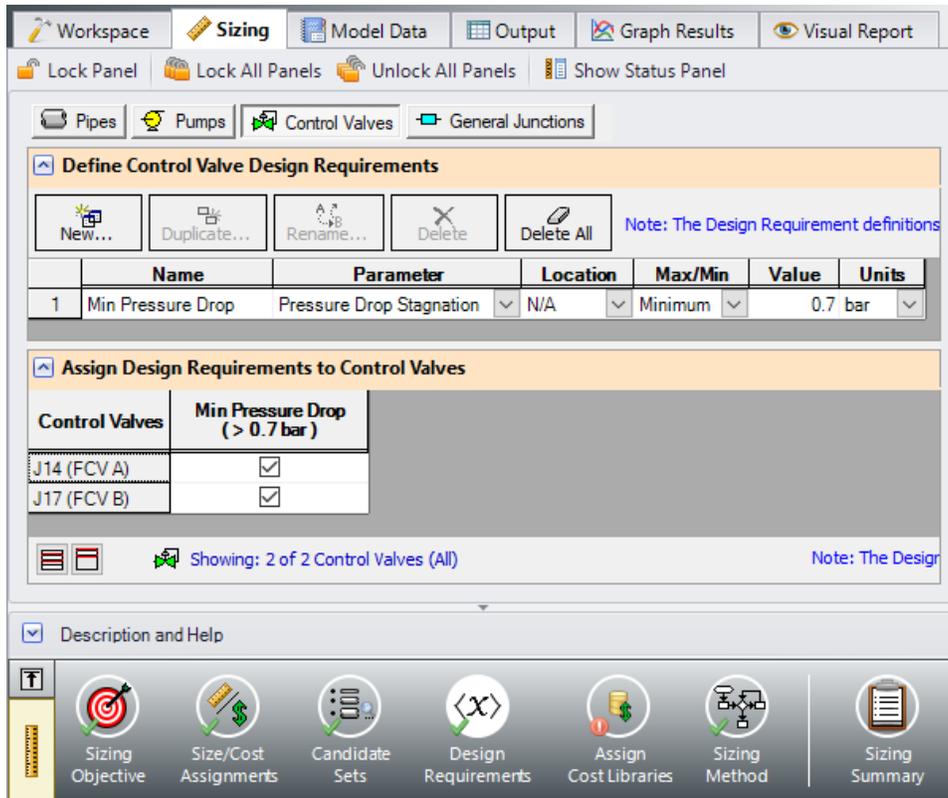


Figure 5: Control Valve Design Requirements

E. Assign Cost Libraries

Select the Assign Cost Libraries button. For this model the engineering and cost libraries have already been created, but we will need to check that they have been connected and applied properly.

➤ **Open the Library Manager** by clicking the Library Manager button to see which libraries are connected. The AFT INTERNAL LIBRARY and LOCAL USER LIBRARY should be available and connected. The AFT internal library also has a cost library associated with it. This example uses the engineering library *Hot Water System.dat* and two cost libraries, *Hot Water System Costs.cst* and *Steel - ANSI Pipe Costs.cst* which are linked with the Hot Water System engineering library and the pipe material library for Steel - ANSI, respectively. If the libraries are connected, there will be a check in the box next to their name, as shown in Figure 6.

If they are not in the available library list, add them from the Examples folder in the AFT Fathom 13 folder. Click Add Existing Library to navigate to and select the library *Hot Water System.dat*. Next, click Add Existing Cost Library and add *Hot Water System Costs.cst* and *Steel - ANSI Pipe Costs.cst*.

The Metric - ANS Hot Water Heat Exchanger junction will only be used for the heat exchanger resistance curve. The Metric - ANS Hot Water Pump junction will be used for the three pumps. These are the only junctions in this library that will be used in this example.

Once they have been added to the list, check to see that they are also connected. If they are not automatically connected, click the check box next to their name to connect them as seen in Figure 6. Click Close.

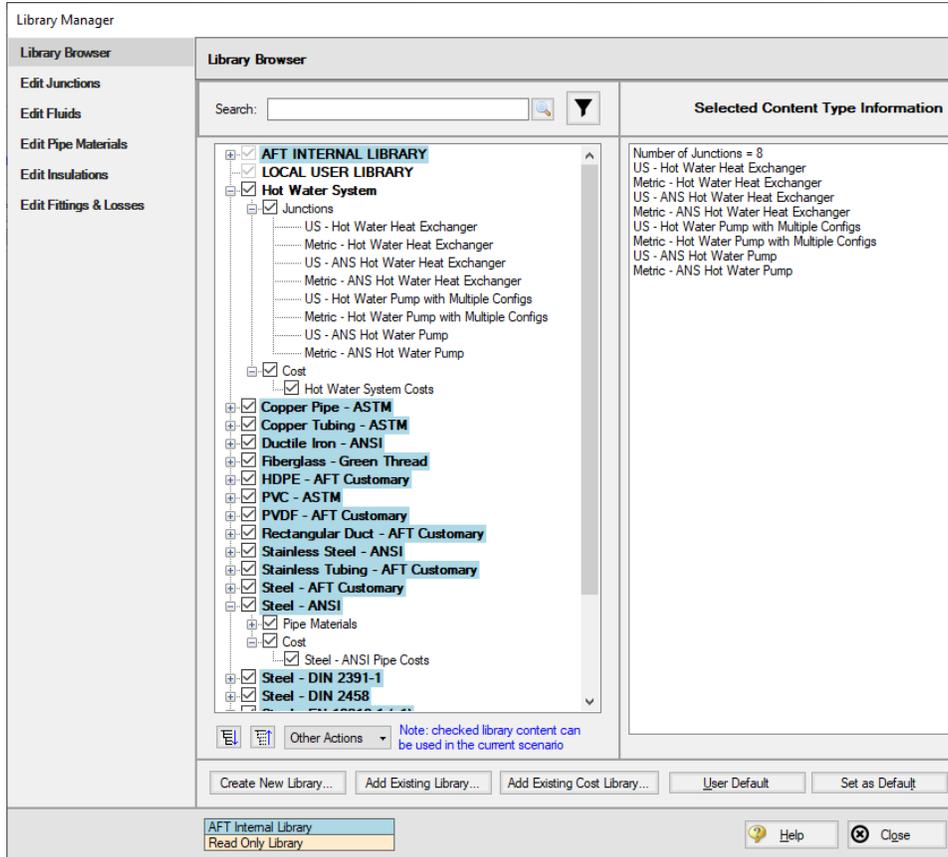


Figure 6: Library Manager with Hot Water Pumps connected

Back in the Assign Cost Libraries window for the Pipes, the **STD Steel Pipe 1"-36"** Candidate Set should now show the **Steel - ANSI Pipe Costs** library applied to it.

Navigate to the Workspace and connect all three pump junctions to the **Metric - ANS Hot Water Pump** library junction by opening the pump properties window and selecting **Metric - ANS Hot Water Pump** from the Library Jct list. Do the same for both Heat Exchanger junctions, selecting **Metric - ANS Hot Water Heat Exchanger**.

Back in the Sizing window on the Assign Cost Libraries panel, click on the Pumps button to view the cost library assignments for the Pumps. Similar to the Pipes, the **Hot Water System Costs** library should be selected for Pumps 4, 7, and 10, showing that it is being applied.

F. Sizing Method

Select the Sizing Method button from the Sizing Navigation panel.

Make sure that Discrete Sizing is selected for the Sizing Type.

For this model both the **Modified Method of Feasible Directions (MMFD)** and the Sequential Quadratic Programming (SQP) methods will produce the same results, but MMFD is much faster. Select MMFD for the Search Method.

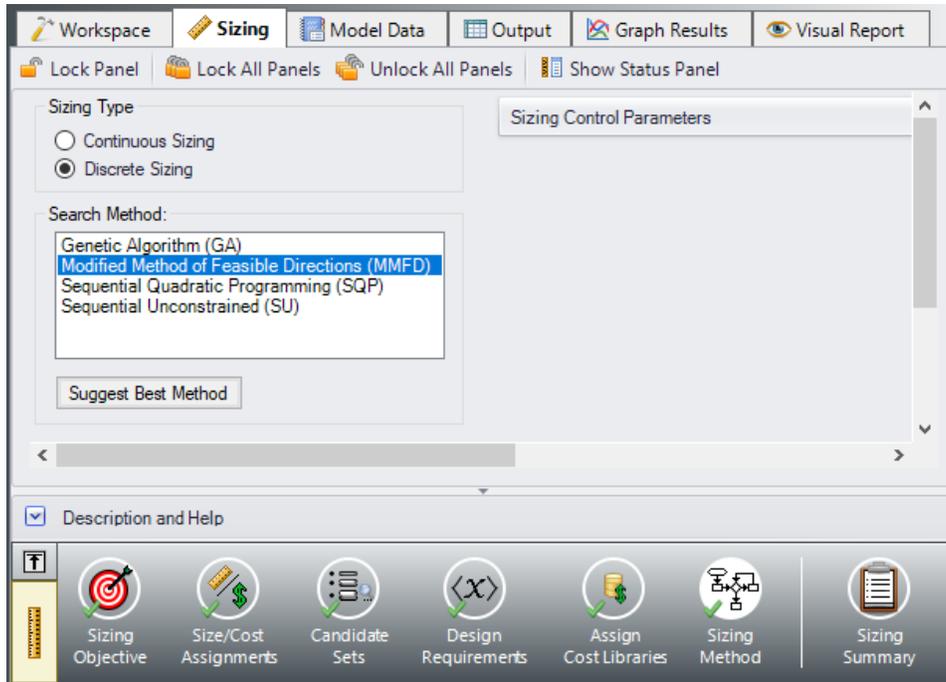


Figure 7: Sizing Method Settings to discretely size the system

Step 6. Create Scenarios

For this system we would like to get a baseline cost for the current design to compare against the sizing results.

In the Scenario Manager in the Quick Access Panel create two scenarios and name them *Before Sizing* and *After Sizing*.

Load the Before Sizing scenario if it is not already active by double-clicking the scenario name in the Scenario Manager. We need to set this scenario to calculate costs without sizing the system.

In the Sizing window go to the Sizing Objective panel, then select **Calculate Cost, Do Not Size**.

Step 7. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 8. Review the Results

The Cost Report for the initial design can be seen in Figure 8. The total cost comes out to \$2,132,398. In the pipe output section there is a Pipe Design Requirements tab and in the junction output section there is a Pump Design Requirements and Control Valve Design Requirements tab. These tabs will show if the current model fulfills the Design Requirements. We can investigate the design requirement tabs and see that the initial system meets all of the Design Requirements, **meaning that it is a feasible solution**; however, a lower cost system can likely be found by sizing the system.

⌵ General Warnings Design Alerts Cost Report Pump Summary Valve Summary Heat Exchanger Summary Reservoir Summary									
Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL
TOTAL OF ALL MODEL COSTS									2,132,398
Total of All Shown Costs			393,463	192,331	585,794	0	1,546,604	1,546,604	2,132,398
Items In Sizing			0	0	0	0	0	0	0
Items Not In Sizing			393,463	192,331	585,794	0	1,546,604	1,546,604	2,132,398
Pipe Subtotal			366,692	184,716	551,408	0	0	0	551,408
	P1	Pipe	Pipe	18,183	9,159	27,343	0	0	27,343
o	P2	Pipe	Pipe	54,549	27,478	82,028	0	0	82,028
o	P3	Pipe	Pipe	9,092	4,580	13,671	0	0	13,671
o	P4	Pipe	Pipe	18,183	9,159	27,343	0	0	27,343
o	P5	Pipe	Pipe	18,183	9,159	27,343	0	0	27,343
o	P6	Pipe	Pipe	9,092	4,580	13,671	0	0	13,671
o	P7	Pipe	Pipe	18,183	9,159	27,343	0	0	27,343
o	P8	Pipe	Pipe	9,092	4,580	13,671	0	0	13,671
o	P9	Pipe	Pipe	18,183	9,159	27,343	0	0	27,343
o	P10	Pipe	Pipe	9,092	4,580	13,671	0	0	13,671
o	P11	Pipe	Pipe	6,061	3,053	9,114	0	0	9,114
o	P12	Pipe	Pipe	6,061	3,053	9,114	0	0	9,114
o	P13	Pipe	Pipe	6,061	3,053	9,114	0	0	9,114
o	P14	Pipe	Pipe	6,061	3,053	9,114	0	0	9,114
o	P15	Pipe	Pipe	9,092	4,580	13,671	0	0	13,671
o	P16	Pipe	Pipe	18,183	9,159	27,343	0	0	27,343
o	P17	Pipe	Pipe	18,183	9,159	27,343	0	0	27,343
o	P18	Pipe	Pipe	9,092	4,580	13,671	0	0	13,671
o	P19	Pipe	Pipe	30,305	15,266	45,571	0	0	45,571
o	P20	Pipe	Pipe	9,092	4,580	13,671	0	0	13,671
o	P21	Pipe	Pipe	18,183	9,159	27,343	0	0	27,343
o	P22	Pipe	Pipe	9,092	4,580	13,671	0	0	13,671
o	P23	Pipe	Pipe	30,305	15,266	45,571	0	0	45,571
o	P24	Pipe	Pipe	9,092	4,580	13,671	0	0	13,671
Pump Subtotal			26,771	7,616	34,386	0	1,546,604	1,546,604	1,580,990
o	J4	Pump	Pump A	8,929	2,540	11,470	0	516,056	527,526
o	J7	Pump	Pump B	8,912	2,535	11,447	0	514,491	525,938
o	J10	Pump	Pump C	8,929	2,540	11,470	0	516,056	527,526

Figure 8: Cost Report for the initial system design before sizing

Step 9. Run the Model and View Results

Switch to the After Sizing scenario in the Scenario Manager. This scenario should already be fully defined for sizing.

Select Run from the Analysis menu to perform the automated sizing calculations. Using the MMFD method, the automated sizing takes about 2 minutes, depending on your machine. When the run has completed click Output to view the results, as shown in Figure 9.

The automated sizing has given us savings of about \$183,000 overall on material, installation, and operation costs over the lifespan of the system. This is a 9% reduction.

Hot Water System - ANS

General Warnings Design Alerts Cost Report Pump Summary Valve Summary Heat Exchanger Summary Reservoir Summary									
Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL
TOTAL OF ALL MODEL COSTS									1,949,264
Total of All Shown Costs			252,323	150,296	402,620	0	1,546,644	1,546,644	1,949,264
Items In Sizing			234,140	141,137	375,277	0	1,546,644	1,546,644	1,921,921
Items Not In Sizing			18,183	9,159	27,343	0	0	0	27,343
Pipe Subtotal			225,552	142,680	368,233	0	0	0	368,233
o	P1	Pipe	Pipe	18,183	9,159	27,343	0	0	27,343
o	P2	Pipe	Pipe	45,112	24,963	70,075	0	0	70,075
o	P3	Pipe	Pipe	7,519	4,160	11,679	0	0	11,679
o	P4	Pipe	Pipe	15,037	8,321	23,358	0	0	23,358
o	P5	Pipe	Pipe	7,431	5,846	13,278	0	0	13,278
o	P6	Pipe	Pipe	3,716	2,923	6,639	0	0	6,639
o	P7	Pipe	Pipe	7,431	5,846	13,278	0	0	13,278
o	P8	Pipe	Pipe	3,716	2,923	6,639	0	0	6,639
o	P9	Pipe	Pipe	7,431	5,846	13,278	0	0	13,278
o	P10	Pipe	Pipe	3,716	2,923	6,639	0	0	6,639
o	P11	Pipe	Pipe	4,062	2,490	6,552	0	0	6,552
o	P12	Pipe	Pipe	2,477	1,949	4,426	0	0	4,426
o	P13	Pipe	Pipe	2,477	1,949	4,426	0	0	4,426
o	P14	Pipe	Pipe	4,062	2,490	6,552	0	0	6,552
o	P15	Pipe	Pipe	7,519	4,160	11,679	0	0	11,679
o	P16	Pipe	Pipe	15,037	8,321	23,358	0	0	23,358
o	P17	Pipe	Pipe	9,631	6,601	16,232	0	0	16,232
o	P18	Pipe	Pipe	4,815	3,301	8,116	0	0	8,116
o	P19	Pipe	Pipe	16,052	11,002	27,054	0	0	27,054
o	P20	Pipe	Pipe	4,815	3,301	8,116	0	0	8,116
o	P21	Pipe	Pipe	9,631	6,601	16,232	0	0	16,232
o	P22	Pipe	Pipe	4,815	3,301	8,116	0	0	8,116
o	P23	Pipe	Pipe	16,052	11,002	27,054	0	0	27,054
o	P24	Pipe	Pipe	4,815	3,301	8,116	0	0	8,116
Pump Subtotal			26,771	7,616	34,387	0	1,546,644	1,546,644	1,581,031
o	J4	Pump	Pump A	8,899	2,531	11,430	0	513,362	524,792
o	J7	Pump	Pump B	8,973	2,554	11,527	0	519,921	531,448
o	J10	Pump	Pump C	8,899	2,531	11,430	0	513,362	524,792

Figure 9: Sizing output based on the initial cost of the system with all pumps operating

Step 10. Create Dependent Design Case

We also want the system to function with one pump turned off. To size a system with two different operating cases, we can create a Dependent Design Case that has the exact same pipes and junctions as the primary design but has a different set of requirements.

Create a new child of the After Sizing scenario in the Scenario Manager, and name it *Sizing with Pump A off*.

Here are the steps to create this Dependent Design Case:

1. Go to the **Sizing Objective** panel and check the box next to **Enable Dependent Design Cases** option.
2. Go to the **Dependent Design Cases** panel. You should now see instructions displayed to create the Dependent Design Case, along with a summary table of dependent design settings. Now go to the **Workspace** and choose **Select All** from the **Edit** menu.
3. Open **Duplicate Special** from **Edit** menu, enter an increment of **100** and select **Make Dependent Design Case**.
4. Click **OK**, then **paste** the duplicated model so that it does not overlap the original.
5. When the Workspace has been duplicated, select the **Pump A** from the **Dependent Design Case**

(J104). In the Pump Properties window select the **Optional tab** and check the **Pump Off No Flow** radio button under Special Conditions. This will turn Pump A off.

We have now made all of the changes required in the Workspace. We will also need to make some adjustments to the Design Requirements for our Dependent Design Case. With Pump A turned off we no longer require any of the pumps to operate between 80% and 100% of BEP. We will therefore need to remove the Pump Design Requirements.

➤ **Go to the Design Requirements panel** and select the **Pumps** button. In the Assign Design Requirements to Pumps section, **remove** the Design Requirements for all of the dependent design pumps by unchecking the boxes as is shown in Figure 10.

Once this is done run the model again and go to the Output window.

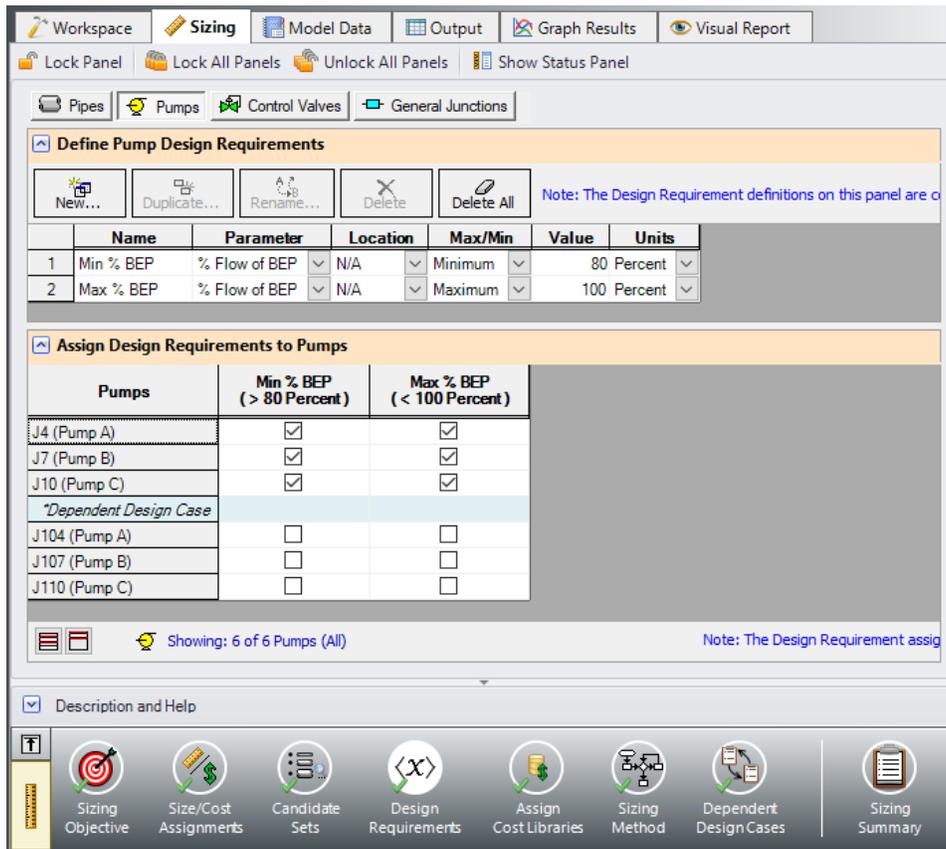


Figure 10: Pump Design Requirements defined for the Dependent Design Case pumps

Summary

Table 1 compares the results for each of the scenarios. The final design is the lowest cost that meets all the requirements. It can be seen that in order to meet the system requirements for the case where Pump A is turned off, slightly larger pipe sizes are used, resulting in a higher system cost.

The Final Design provides a \$165,412 (8%) cost reduction from the Initial Design while still meeting all design requirements.

Table 1: Summary of costs for Hot Water System scenarios

Design Stage	Initial Design	Design	Final Design
Scenario	Before Sizing	After Sizing	Sizing with Pump A off
Meets all Requirements	Yes	Yes	Yes
Material	393,463	252,323	265,590
Installation	192,331	150,296	154,765
Non-Recurring Sub Total	585,794	402,620	420,355
Operation/Energy	1,546,604	1,546,644	1,546,631
TOTAL	2,132,398	1,949,264	1,966,986

Variable Drive - ANS

Summary

As seen in other examples, the ANS module can be used to size a system for life cycle cost. The ANS module will size the complete system, pumps and pipes to minimize the total cost of operating the system for the life of the system. Often sizing the pumps and pipes is not enough because the control system is often a large portion of the initial cost and the method of control, flow control valves (FCV) or variable frequency drives (VFD), can have a large impact on the system energy usage. This example will demonstrate how to size pumps and control valves in parallel and then will investigate the savings of using a VFD to control flow instead of a FCV.

Note: This example can only be run if you have a license for the ANS module.

Topics Covered

- Using Scenarios to size a system for two different sets of system requirements
- Using Common Size Groups and observing their effect on how well the ANS module can size a system

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the [Beginner - Three-Reservoir - ANS](#) example, and is familiar with the basics of ANS analysis.

Since this example employs many concepts from the [Cooling System - ANS](#) example, it is recommended to complete that example first before beginning this example.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- ***Metric - Variable Drive - ANS.fth***
- ***Variable Drive.dat*** - engineering library
- ***Variable Drive - Pump Costs.cst*** - cost library for ***Variable Drive.dat***

- **Variable Drive - VFD Costs.cst** - cost library for **Variable Drive.dat**
- **Variable Drive - Pipe Costs.cst** - cost library for **Steel - ANSI** pipes

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Set up Library Manager

This example uses engineering and cost libraries. It is important that these libraries are added and connected to Fathom for defining the pipes and junction.

Go to the Library menu and open Library Manager. Click Add Existing Library and navigate to the AFT Fathom 13 Examples folder. Select the library **Variable Drive.dat** and click Open. Make sure that the Variable Drive engineering library in Library Manager has a check mark next to it. The cost libraries will be added later during the Sizing steps.

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate ANS and select Network to enable the ANS module for use. A new group will appear in Analysis Setup titled Automatic Sizing. Click OK to save the changes and exit Analysis Setup. A new Primary Window tab will appear between Workspace and Model Data titled Sizing. Open the Analysis menu to see the new option called Automatic Sizing. From here you can quickly toggle between Not Used mode (normal AFT Fathom) and Network (ANS mode).

Step 4. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = User Specified Fluid
 - b. Name = Lube Oil
 - c. Density = 0.7 S.G. water
 - d. Dynamic Viscosity = 0.29 kg/sec-m

Step 5. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions

fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

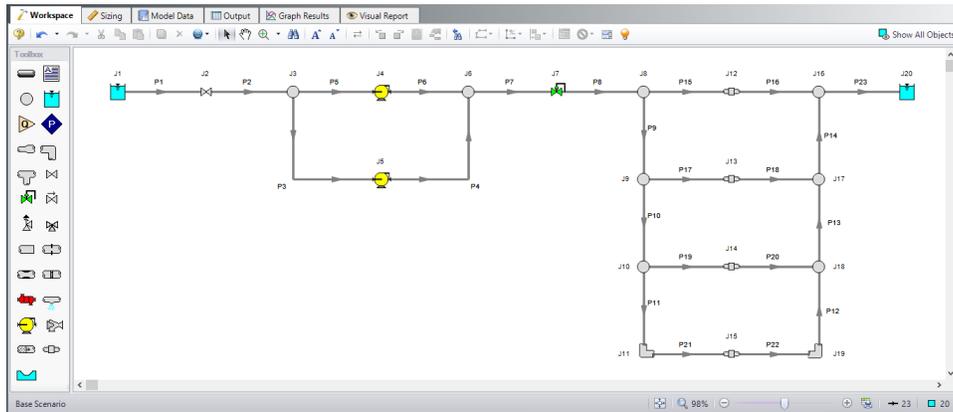


Figure 1: Layout of Generator Lube Oil System

Pipe Properties

1. Pipe Model tab
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 6 inch
 - d. Type = schedule 40
 - e. Friction Model Data Set = Standard
 - f. Lengths =

Pipe	Length (meters)
1	6
2-4	3
5-6	2
7	1
8	5
9-14	3
15-22	6
23	15

Junction Properties

1. J1 Reservoir
 - a. Name = Lube Oil Supply Tank
 - b. Liquid Surface Elevation = 6 meters
 - c. Liquid Surface Pressure = 1 atm
 - d. Pipe Depth = 3 meters
2. J2 Valve
 - a. Name = Isolation Valve
 - b. Inlet Elevation = 0 meters
 - c. Valve Data Source = User Specified
 - d. Loss Model = K Factor
 - e. K = 0.15
3. All Branches
 - a. Elevation = 0 meters
4. J4 & J5 Pumps
 - a. J4 Name = Primary Pump
 - b. J5 Name = Auxiliary Pump
 - c. Library Jct = Pump 450 m3/hr Sizing
 - d. Inlet Elevation = 0 meters
 - e. J5 Special Condition = Pump Off No Flow
5. J7 Control Valve
 - a. Name = 450 m3/hr Control
 - b. Inlet Elevation = 0 meters
 - c. Valve Type = Constant Pressure Drop (PDCV)
 - d. Control Setpoint = Pressure Loss
 - e. Pressure Drop = 70 kPa
6. J11 & J19 Bend
 1. Inlet Elevation = 0 meters
 2. Type = Standard Elbow (knee, threaded)
 3. Angle = 90 Degrees
7. J12-J15 General Components
 - a. J12 Name = Generator #1
 - b. J13 Name = Generator #2
 - c. J14 Name = Generator #3
 - d. J15 Name = Generator #4
 - e. Library Jct = Metric - Generator
 - f. Inlet Elevation = 0 meters
8. J20 Reservoir
 - a. Name = Return Tank
 - b. Liquid Surface Elevation = 5 meters
 - c. Liquid Surface Pressure = 1 atm
 - d. Pipe Depth = 5 meters

The system supplies lube oil to a bank of four generators. Each generator requires a flow rate of 115 m³/hr. The auxiliary pump will not be turned on but the material and installation pump costs will be included in the cost report and automated sizing.

The system uses the following specifications:

1. Minimum 70 kPa drop across the flow control valve
2. Flow rate at each generator must be between 100 and 125 m³/hr
3. The cost of the auxiliary pump and piping must be included in the sizing.
4. The pump is expected to be about 80% efficient.
5. The system uses lube oil with a specific gravity of 0.7 and viscosity of 0.29 kg/s-m.

Step 6. Configure Sizing Settings

We want to first size our system to determine the required pump parameters, which will then be used to choose a pump model and perform a final sizing calculation.

To start, set up the model to size the pump system for life cycle cost. Select the Sizing tab.

A. Sizing Objective

The Sizing Objective panel should be selected by default from the navigation panel. Choose the settings as follows:

For the Sizing Option choose **Perform Sizing**.

For the Objective, choose **Monetary Cost**, and select the **Minimize** option from the drop-down list.

Under Options to Minimize Monetary Cost, choose **Size for Initial and Recurring Costs**. This should automatically configure the Cost Objective table to include the material, installation, and energy costs in the sizing.

Define a **10 year** System Life.

For the Energy Cost area choose **Use This Energy Cost Information**, with a cost of **0.12 U.S. Dollars Per kW-hr**.

B. Size/Cost Assignments

On the Sizing Navigation panel go to the **Size/Cost Assignments** panel.

We are designing a new system, so we want to include all of the pipes in the model for sizing.

➤ **Select the All button** at the bottom of the panel to select all pipes, then click **Always Include in Cost** at the top of the table to move all pipes to be included in the sizing calculation.

To decrease the analysis speed and reduce complexity, it is useful to **create Common Size Groups** to group the pipes in different sections of the model. For this system we want to create six groups to be sized, as can be seen in Figure 2 based on the pipe coloring, and Figure 3 based on the table. The groups were chosen by considering sections of pipe which would logically have similar demands due to the flow capacity and connected junctions.

Create each of the groups by selecting the appropriate pipes in the Workspace, then right-clicking on the Workspace and choosing Add Pipes to Common Size Group -> New Group and giving the group a name. Alternatively the groups can be created on the Sizing window by clicking New to create and name the group, then adding pipes to the group using the radio buttons. The Size/Cost Assignments panel should appear as shown in Figure 3.

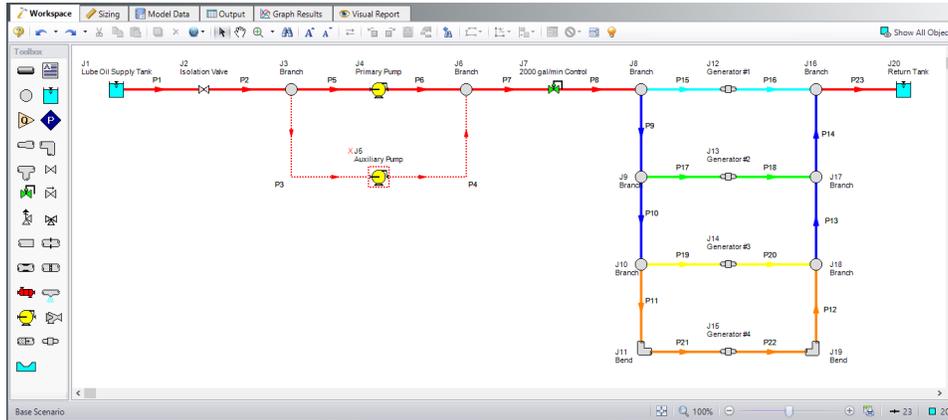


Figure 2: Workspace showing pipe groups by color

Pipes	Sizing and Cost Options				Not in Group	Pipe Grouping					
	Automatically Size		Do Not Size			Main Lines	Headers	Common Size Groups			
	Always Include in Cost	Include in Cost if Size Changes	Include in Cost	Do Not Include in Cost				Generator #1	Generator #2	Generator #3	Generator #4
P1	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P2	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P3	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P4	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P5	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P6	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P7	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P8	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P9	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P10	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P11	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P12	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P13	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P14	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P15	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P16	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P17	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P18	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P19	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P20	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P21	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P22	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	
P23	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>	

Figure 3: Size/Cost Assignments using Common Size Groups to simplify calculations

➤ **Click the Pumps button** at the top of the panel to switch to the Pump Size and Cost Assignments. As stated earlier, we want to size both pumps, even though the Auxiliary Pump is not in use, so select **Include in Cost Report and Sizing** for each of the Pump junctions. Since the pumps are in parallel in this model, it is desired for the pumps to be the same size. This means that they will have the same material and installation costs, though the maintenance cost will be different since the Auxiliary Pump is not regularly used, unlike the Primary Pump. To account for this we will need to create a Maximum Cost Group, which will set both pumps to use the cost information for the most expensive pump in the group. If a Maximum Cost Group is not used, then the Auxiliary Pump will have its material and installation costs set to

the lowest possible cost. (The cost at the pump is estimated using power, and the power at the Auxiliary Pump is set at zero since it is not in use during normal operation.)

➤ **Create a New Maximum Cost Group** named *Pumps*, and add the Material and Installation costs to it for each of the pumps, but under Maintenance, make sure to leave both pumps in the Not in Group column. The Pumps Size/Cost Assignments should appear as shown in Figure 4.

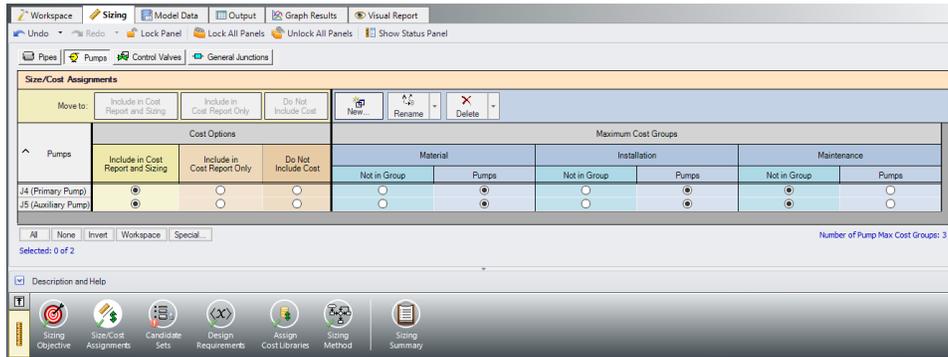


Figure 4: Pump Size/Cost Assignments with Maximum Cost Groups assigned

C. Candidate Sets

Go to the **Candidate Sets** panel.

For this system we will consider a range of Steel-ANSI schedule 40 pipes as candidates. To define this in Fathom, create a new Candidate set as follows:

1. Create a new set by clicking **New**, and name the set *Galvanized Steel Pipe 1-18 in*.
2. Select **Steel - ANSI** as the material from the drop-down list
3. In the Available Material Sizes and Types, expand the sizes for **schedule 40**, then add each of the sizes from **1 inch to 18 inch** by double clicking the names or selecting them and clicking Add>>.
4. Click **OK** to finish editing the set.

In the bottom section of the panel, assign each of the Common Size Groups to the Galvanized Steel Pipe 1-18 in Candidate Set. The panel should appear as shown in Figure 5.

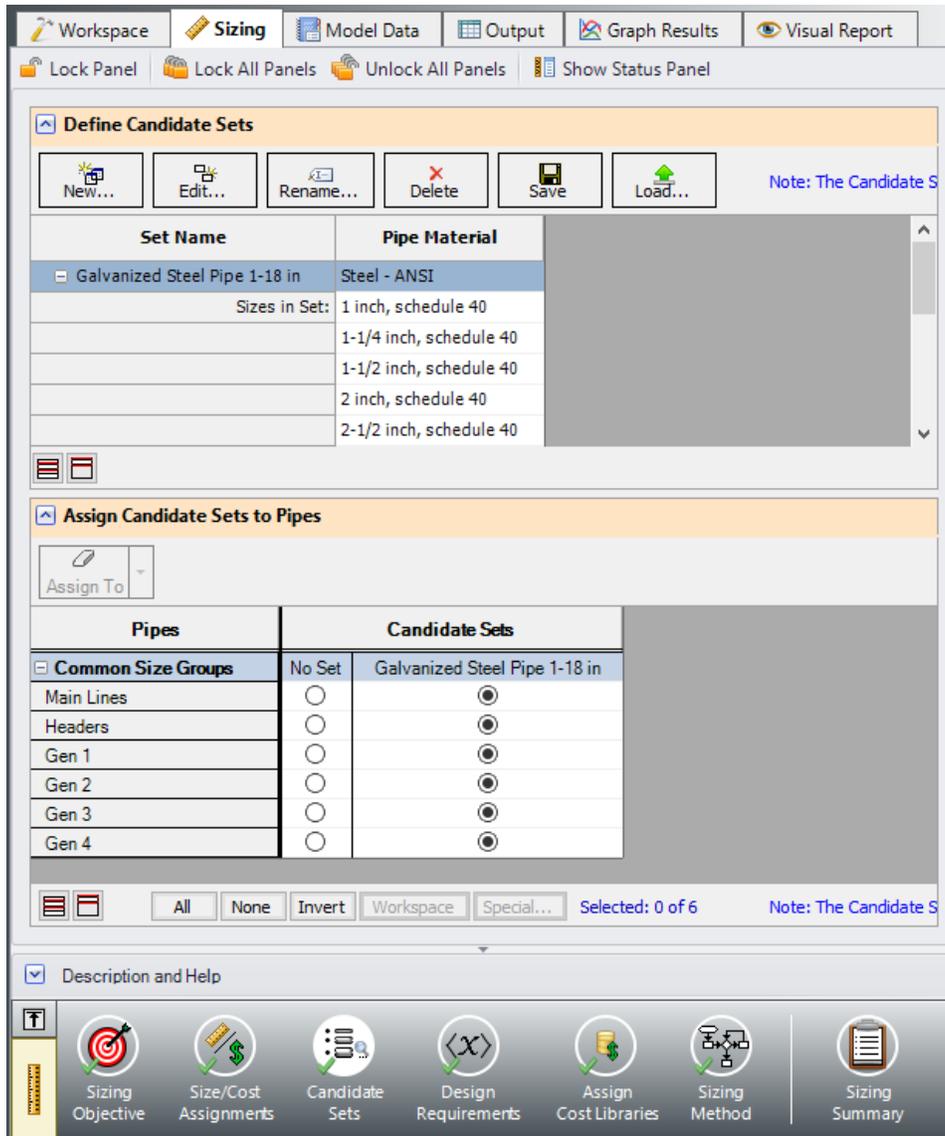


Figure 5: Candidate Sets panel fully defined for the model

D. Design Requirements

Go to the **Design Requirements** panel.

As can be seen above, this model has three given Design Requirements which need to be applied for the generators and control valve. Since we are not able to directly apply a flow rate design requirement on the generators, we will instead apply the flow rate requirements for the generators to the pipes adjacent to the generators. To do this:

1. Make sure the Pipes button at the top of the window is selected.
2. Click **New** under Define Pipe Design Requirements.

3. Enter the name *Maximum Generator Flow*.
4. In the table for Parameter, select **Volumetric Flow Rate**.
5. For Max/Min, choose **Maximum**, and enter **125 m3/hr**.
6. Repeat the process from steps 1-5 to define a Design Requirement for a **Minimum** generator flow rate of **100 m3/hr**.

Now that the generator flow rate requirements are defined, apply them to the common size groups for the generator piping (Gen 1, Gen 2, Gen 3 and Gen 4) by checking the appropriate boxes in the Assign Design Requirements to Pipes section.

Click on the **Control Valves** button and create a **New** requirement named *Minimum Pressure Drop* to set the **Pressure Drop Stagnation** parameter to a **Minimum** of **70 kPa**. Apply the Design Requirement to Control Valve J7.

E. Assign Cost Libraries

Go to the Assign Cost Libraries panel. For this model, the engineering and cost libraries have already been created, but we will need to check that they have been connected and applied properly.

➤ **Open the Library Manager**, and check that the engineering library **Variable Drive** is connected, indicated by a check mark next to its name. If it is not connected, then check the box next to its name.

Next, connect to the three cost libraries. Click Add Existing Cost Library and navigate to the AFT Fathom 13 Examples folder and add the cost libraries **Variable Drive - Pump Costs.cst**, **Variable Drive - VFD Costs.cst**, and **Variable Drive - Pipe Costs.cst**. The two cost libraries, Variable Drive - Pump Costs and Variable Drive - VFD Costs should be shown indented below the engineering library Variable Drive, and the Variable Drive - Pipe Costs library should be indented below Steel - ANSI. Once all of the necessary libraries have been connected, click **OK**.

In the Assign Cost Libraries panel for the Pipes, All should be selected for the Galvanized Steel Pipe 1-18 in Candidate Set, with the Variable Drive - Pipe Costs library being applied.

➤ **Click on the Pumps button** to view the cost library assignments for the Pumps. We will only be using Variable Drive - Pump Costs for this scenario, so check the box for this cost library for both J4 and J5 to make sure that we are not applying any extra costs.

F. Sizing Method

Select the Sizing Method button to go to the corresponding panel.

➤ **Choose Discrete Sizing**, since it is desired to select discrete sizes for each of the pipes in the model.

This model contains six design variables and 18 design requirements, so the best methods to choose from will be the MMFD or SQP method.

➤ **Select the default method**, Modified Method of Feasible Directions (MMFD).

Step 7. Create Scenarios

Before running the model, create two child scenarios, one named *Calculate Costs, Do Not Size* and the other named *Flow Control Valve Initial Cost Sizing*. This will provide a comparison of the initial cost of the model without sizing, to the cost of the sized system.

Load the Calculate Costs, Do Not Size scenario and go to the Sizing window, then to the Sizing Objective panel. Select Calculate Costs, Do Not Size. We will run this scenario first.

Step 8. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 9. Review the Not Sized and Initial Cost Sizing results

With all the pipes set to 6 inches the design requirements are met and the system is in a reasonable state. Navigate to the Cost Report tab and note that the total cost is \$790,488.

Now load the **Flow Control Valve Initial Cost Sizing** scenario and go to the Sizing window to the **Sizing Objective** panel. Under Options to Minimize Monetary Cost, choose **Size for Initial Cost, Show Energy Costs**. The scenario is now ready to be run.

Run the model, then click the Output button once the automated sizing is completed.

After sizing the model for Initial Cost, the total cost over 10 years is now calculated as \$728,970. The new pipe sizes and cost totals can be seen in Figure 6 below.

Although this sized system uses larger pipe sizes, and therefore increased material and installation cost, than the initial design (4-8 inch versus only 6 inch), the savings comes from a the sized pump requiring much less power to operate. In one short simulation, the system was sized to meet all design requirements and reduce the cost by close to \$60,000.

Variable Drive - ANS

General Warnings Design Alerts Cost Report Pump Summary Valve Summary Reservoir Summary										
Table Units: U.S. Dollars	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL	
TOTAL OF ALL MODEL COSTS										728,970
Total of All Shown Costs			33,148	12,698	45,846	0	683,124	683,124	728,970	
Items In Sizing			33,148	12,698	45,846	0	0	0	45,846	
Items Not In Sizing			0	0	0	0	683,124	683,124	683,124	
Pipe Subtotal			11,522	6,440	17,962	0	0	0	17,962	
o P1	Pipe	Pipe	1,004	492	1,496	0	0	0	1,496	
o P2	Pipe	Pipe	502	246	748	0	0	0	748	
o P3	Pipe	Pipe	502	246	748	0	0	0	748	
o P4	Pipe	Pipe	502	246	748	0	0	0	748	
o P5	Pipe	Pipe	335	164	499	0	0	0	499	
o P6	Pipe	Pipe	335	164	499	0	0	0	499	
o P7	Pipe	Pipe	167	82	249	0	0	0	249	

Pipes Sizing Pipe Design Requirements Library Sources															
Pipe	Name	Vol. Flow Rate (m3/hr)	Velocity (meters/sec)	P Static Max (bar)	P Static Min (bar)	Elevation Inlet (meters)	Elevation Outlet (meters)	dP Stag. Total (bar)	dP Static Total (bar)	dP Gravity (bar)	dH (meters)	P Static In (bar)	P Static Out (bar)	P Stag. In (bar)	P Stag. Out (bar)
1	Pipe	450.0	3.873	1.320	1.167	3.000	0	-0.153290	-0.153290	-0.2058	0.7651	1.167	1.320	1.219	1.372
2	Pipe	450.0	3.873	1.312	1.286	0.000	0	0.026238	0.026238	0.0000	0.3825	1.312	1.286	1.364	1.338
3	Pipe	0.0	0.000	1.338	1.338	0.000	0	0.000000	0.000000	0.0000	0.0000	1.338	1.338	1.338	1.338
4	Pipe	0.0	0.000	5.465	5.465	0.000	0	0.000000	0.000000	0.0000	0.0000	5.465	5.465	5.465	5.465
5	Pipe	450.0	3.873	1.286	1.268	0.000	0	0.017492	0.017492	0.0000	0.2550	1.286	1.268	1.338	1.321
6	Pipe	450.0	3.873	5.430	5.413	0.000	0	0.017492	0.017492	0.0000	0.2550	5.430	5.413	5.483	5.465
7	Pipe	450.0	3.873	5.413	5.404	0.000	0	0.008746	0.008746	0.0000	0.1275	5.413	5.404	5.465	5.456
8	Pipe	450.0	3.873	4.704	4.660	0.000	0	0.043729	0.043729	0.0000	0.6376	4.704	4.660	4.756	4.713
9	Pipe	333.8	4.975	4.626	4.568	0.000	0	0.058362	0.058362	0.0000	0.8509	4.626	4.568	4.713	4.654
10	Pipe	219.9	3.277	4.617	4.578	0.000	0	0.038447	0.038447	0.0000	0.5605	4.617	4.578	4.654	4.616
11	Pipe	107.5	3.636	4.570	4.473	0.000	0	0.096808	0.096808	0.0000	1.4114	4.570	4.473	4.616	4.519

Figure 6: Pipe Sizes based on automated sizing for Initial Costs

Step 10. Size the system based on lifetime cost

In the Scenario Manager on the Quick Access Panel, make a child scenario from the Base Scenario named *Flow Control Valve Life Cycle Sizing*. Load then run this scenario.

On the Output tab in the Cost Report, it can be seen that the total cost for this case is \$613,160 using pipe sizes ranging from 8 inches to 12 inches, as can be seen in Figure 7. Though not shown, the Pump section of the Cost Report shows that the two pumps have been set to the same Material and Installation costs due to the Maximum Cost Group that was defined on the Size/Cost Assignments panel. Since the Initial Cost sizing results in a lifetime cost *increase* of about \$50,000, the Life Cycle sizing results will be used for the pump configuration information to choose a pump from a manufacturer.

General Warnings Design Alerts Cost Report Pump Summary Valve Summary Reservoir Summary						
	Table Units: U.S. Dollars	Material	Installation	Non-Recurring Sub Total	Operation/Energy	TOTAL
TOTAL OF ALL MODEL COSTS						336,280
Total of All Shown Costs	43,237	15,863	15,863	59,100	277,180	336,280
Items In Sizing	43,237	15,863	15,863	59,100	277,180	336,280
Items Not In Sizing	0	0	0	0	0	0
Pipe Subtotal	24,514	10,512	10,512	35,026	0	35,026
o P1	1,870	738	738	2,608	0	2,608
o P2	935	369	369	1,304	0	1,304
o P3	623	246	246	869	0	869
o P4	623	246	246	869	0	869
o P5	935	369	369	1,304	0	1,304
o P6	935	369	369	1,304	0	1,304
o P7	312	123	123	435	0	435
o P8	1,558	615	615	2,174	0	2,174
o P9	4,675	1,845	1,845	6,521	0	6,521
o P10	753	290	290	1,043	0	1,043
o P11	753	290	290	1,043	0	1,043
o P12	753	290	290	1,043	0	1,043
o P13	753	290	290	1,043	0	1,043

Pipes Sizing Pipe Design Requirements Library Sources					
Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (inches)
1	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
2	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
3	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
4	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
5	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
6	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
7	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
8	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
9	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
10	Pipe	Steel - ANSI	10 inch	schedule 40	10.020
11	Pipe	Steel - ANSI	10 inch	schedule 40	10.020
12	Pipe	Steel - ANSI	10 inch	schedule 40	10.020
13	Pipe	Steel - ANSI	10 inch	schedule 40	10.020
14	Pipe	Steel - ANSI	8 inch	schedule 40	7.981
15	Pipe	Steel - ANSI	8 inch	schedule 40	7.981
16	Pipe	Steel - ANSI	8 inch	schedule 40	7.981

Figure 7: Pipe Sizes based on automated sizing for Life Cycle Costs

Step 11. Choose an Actual Pump for the System

Create a child scenario from the Flow Control Valve Life Cycle Sizing scenario, and name the new scenario Flow Control Valve *Real Pump*. We will now need to update the Pumps and Control Valve with the settings for the proposed system. Load the new Flow Control Valve Real Pump scenario using the Scenario Manager, then make the following changes:

Variable Drive - ANS

1. For each of the pumps open the Pump Properties window and select **Pump 450 m3/hr Real** from the Library Jct list. This will copy the Pump Curve data into the pump. Click **OK**. Make sure to do the same for the second pump.
2. Open the Control Valve and change the Valve Type to **Flow Control (FCV)** with a Volumetric Flow Rate of 450 m3/hr. Click **OK**

The model should now be ready for analysis. Run the model once more and go to the Output window to review the results, as can be seen in Figure 8 and Figure 9. Note that this scenario does take a bit longer to run than the sizing scenario. With data for the actual pump, the automated sizing finds a slightly more efficient design for the Life Cycle cost analysis by using 6 inch lines at some of the generators rather than 8 inch lines, which provides approximately \$20,000 reduction in cost from the sizing scenario. Note that the Overall Efficiency is 83% for the real pump compared to the assumed 80% of the original sizing pump.

Jct	Results Diagram	Name	Vol. Flow (m3/hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (Kw)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
4	Show	Primary Pump	450.0	87.43	3.361	49.00	83.15	100.0	50.49	444.2	101.3	N/A	1.700
X5	Show	Auxiliary Pump	0.0	0.00	N/A	N/A	N/A	0.0	N/A	N/A	N/A	N/A	N/A

Figure 8: Pump Summary for operation of actual pump

Table Units: U.S. Dollars										
	Type	Name	Material	Installation	Non-Recurring Sub Total	Maintenance	Operation/Energy	Recurring Sub Total	TOTAL	
TOTAL OF ALL MODEL COSTS										592,908
Total of All Shown Costs			46,109	16,073	62,182	0	530,726	530,726	592,908	
Items In Sizing			46,109	16,073	62,182	0	530,726	530,726	592,908	
Items Not In Sizing			0	0	0	0	0	0	0	
Pipe Subtotal			26,109	11,073	37,182	0	0	0	37,182	
o P1	Pipe	Pipe	1,870	738	2,608	0	0	0	2,608	
o P2	Pipe	Pipe	935	369	1,304	0	0	0	1,304	
o P3	Pipe	Pipe	935	369	1,304	0	0	0	1,304	
o P4	Pipe	Pipe	935	369	1,304	0	0	0	1,304	
o P5	Pipe	Pipe	623	246	869	0	0	0	869	
o P6	Pipe	Pipe	623	246	869	0	0	0	869	
o P7	Pipe	Pipe	312	123	435	0	0	0	435	

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (inches)
1	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
2	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
3	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
4	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
5	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
6	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
7	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
8	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
9	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
10	Pipe	Steel - ANSI	12 inch	schedule 40	11.938
11	Pipe	Steel - ANSI	8 inch	schedule 40	7.981

Figure 9: Output for automated sizing with actual pump curve entered

Step 12. Compare Cost Using a VFD Pump

As a final step we will compare an alternate VFD model for the pump to analyze the cost savings which can be achieved by using a VFD instead of using a flow control valve. To start, make an additional child scenario from the FCV Life Cycle Sizing scenario, and name it *VFD Pump*. Complete the following steps to update the model:

1. Go to the Workspace. For each of the pumps, open the Pump Properties window and choose the **Pump 450 m3/hr VFD Real** pump from the Library Jct list. Click **OK**
2. Open the Control Valve Properties window. On the Optional tab set the Special Condition to **Fully Open - No Control**. Click **OK**.
3. Go to the Sizing window, and go to the **Design Requirements** panel. For the Control Valve **uncheck** the box for the minimum pressure drop requirement, since the control valve is not active.
4. Navigate to the **Assign Cost Libraries** panel. We will need to represent the additional Material/Installation costs for the VFD, so change the Libraries to **All** so that all libraries will be applied for each of the pumps.

The changes for this scenario should now be complete. Run the model once more and go to the Output window to review the results. With the VFD Pump being used, the sizing for the pipes is actually the same as the FCV Life Cycle Sizing scenario. However, the cost is significantly decreased to \$485,016. This is primarily due to the decreased speed of the pump, as can be seen in Figure 10.

⌕ General Warnings Design Alerts Cost Report Pump Summary Valve Summary Reservoir Summary													
Jct	Results Diagram	Name	Vol. Flow (m3/hr)	Mass Flow (kg/sec)	dP (bar)	dH (meters)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (kW)	BEP (m3/hr)	% of BEP (Percent)	NPSHA (meters)	NPSHR (meters)
4	Show	Primary Pump	450.0	87.43	2.677	39.04	82.71	91.75	40.43	407.5	110.4	N/A	1.498
X5	Show	Auxiliary Pump	0.0	0.00	N/A	N/A	N/A	0.00	N/A	N/A	N/A	N/A	N/A

Figure 10: Pump summary for operation of VFD pump

Summary

A summary of the total costs for each of the scenarios can be seen in Table 1. It can be seen from the table that this particular system over its 10 year life span is heavily weighted towards operational costs; as the operational costs are further decreased the overall cost for the system decreases, even though the initial costs may be increased.

Table 1: Summary of Automated Sizing Results for Lube Oil System in thousands of dollars

Life Cycle Cost (U.S. Dollars)	Feasible?	Material	Installation	Total Non-Recurring	Operation/Energy	Total Life Cycle
Calculate Costs, Do Not Size	Yes	34,598	14,148	48,746	741,742	790,488
Flow Control Valve Initial Cost Sizing	Yes	33,148	12,698	45,846	683,124	728,970
Flow Control Valve Life Cycle Sizing	Yes	44,778	16,407	61,185	551,975	613,160
Flow Control Valve Real Pump	Yes	46,190	16,073	62,182	530,726	592,908
Variable Frequency Drive Pump	Yes	46,190	16,073	62,182	422,834	485,016

Housing Project - ANS

Summary

This example will size the pipes for a water supply system to a housing development. This example has three operating states. These states are the following:

1. No fire hydrants open, supply to each house is 2.5 m³/hr with a minimum pressure of 4.0 barG (400 kPa(g)).
2. North Hydrant open with 20 m³/hr at 6.2 barG (620 kPa(g)), supply to each house is 0.5 m³/hr, no minimum pressure.
3. South Hydrant open with 20 m³/hr at 6.2 barG (620 kPa(g)), supply to each house is 0.5 m³/hr, no minimum pressure.

For this example, we will evaluate all three conditions. First, we will use the ANS module to size the system assuming a requirement to use only one pipe size throughout. After this we will size the pipes assuming the pipes can be three different sizes.

Note: This example can only be run if you have a license for the ANS module.

Topics Covered

- Using Dependent Design Cases to satisfy multiple operating modes for a system
- Using the Scenario Manager to size a system for multiple sets of system requirements
- Understanding the effects of Common Size groups on sizing

Required Knowledge

This example assumes the user has already worked through the Walk-Through Examples section, and has a level of knowledge consistent with the topics covered there. If this is not the case, please review the Walk-Through Examples, beginning with the [Beginner - Three Reservoir](#) example. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Model example.

In addition, it is assumed that the user has worked through the [Beginner - Three-Reservoir - ANS](#) example, and is familiar with the basics of ANS analysis.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- **Metric - Housing Project.fth**
- **Metric - Housing Project - ANS.fth**
- **Housing Project Costs.cst** - cost library for **PVC - ASTM** pipes

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Open the model

Open the **Metric - Housing Project.fth** example file and save it as a new file. We will not need the existing child scenarios for this example: right click on the Base Scenario in the Scenario Manager and select **Delete All Children**. Make sure that the Workspace looks like Figure 1 below:

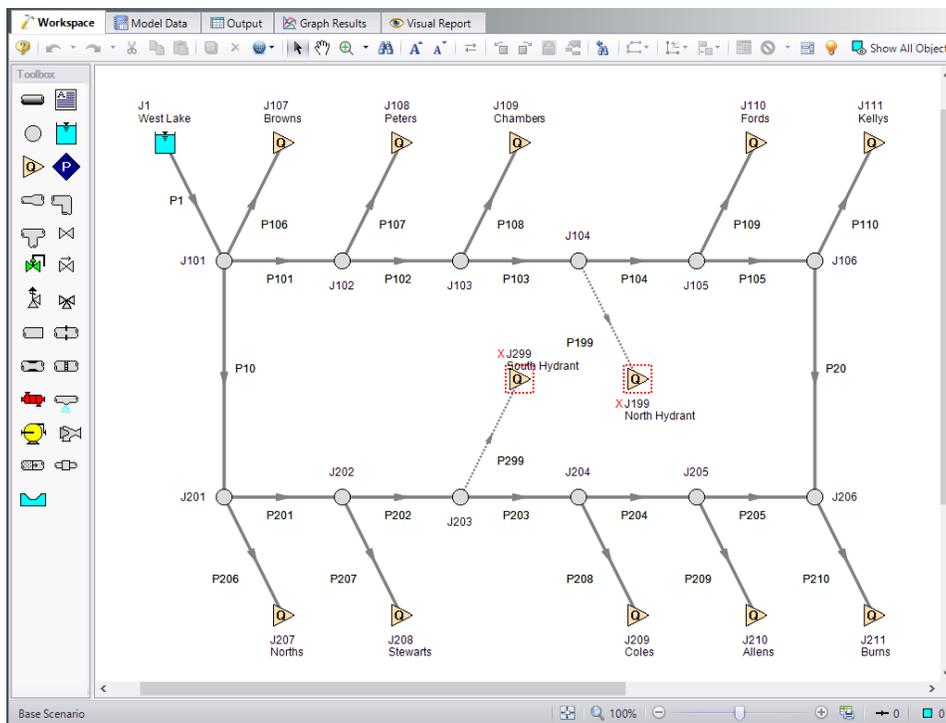


Figure 1: Model layout for normal system operation

This example consists of a reservoir that supplies water to 10 houses and two fire hydrants. All the pipes in the system are Schedule 40 PVC pipe. The supply line to each house is set to 1 inch PVC. The automated sizing of this system will only be used to size the supply line from the reservoir, the neighborhood mains and hydrants.

As mentioned in the introduction, the supply to each house with no fire hydrants open (i.e., the Normal Flow case) is 2.5 m³/hr with a minimum pressure of 4.0 barG (400 kPa(g)). The model can be set up in two ways;

1. Each house could be represented as an assigned flow with a minimum pressure constraint placed on the supply pipe to each house (as done here), or,
2. Each house can be represented as an assigned pressure with a minimum flow constraint placed on the pipe.

The first option was chosen because with either fire hydrant open, there is still an assigned flow on each house but there is no minimum pressure.

Step 3. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate ANS and select Network to enable the ANS module for use. A new group will appear in Analysis Setup titled Automatic Sizing. Click OK to save the changes and exit Analysis Setup. A new Primary Window tab will appear between Workspace and Model Data titled Sizing. Open the Analysis menu to see the new option called Automatic Sizing. From here you can quickly toggle between Not Used mode (normal AFT Fathom) and Network (ANS mode).

Step 4. Configure Sizing Settings

For this system we will be sizing the neighborhood mains for three different operating cases, one for normal operation, and two different cases with hydrants in use. When modeling multiple design cases, the different cases need to be broken up into a primary design case, of which there is one, and dependent design cases, of which there can be any number.

The primary design case can be any of the design cases that are being addressed in your model. Typically, it will represent the primary operating condition of the pipe system. If it is not clear which case should be the primary case, just pick any of the cases and call that the primary one. Design cases other than the primary case are referred to as dependent design cases. This naming convention is used since the dependent design cases are built and defined by duplicating the primary design case, as will be discussed later. The normal flow case will be used as the primary design case, so we will start setting up the sizing settings based on the requirements for normal operation.

A. Sizing Objective

Go to the Sizing window and the Sizing Objective panel should open by default. For this analysis, we are interested in sizing the system considering the monetary cost for the initial costs only.

1. Set the Sizing Option to **Perform Sizing**, if not already.
2. For the Objective, choose **Monetary Cost**, and make sure **Minimize** is selected from the drop-down list.

- Under Options to Minimize Monetary Cost, choose **Size for Initial Cost**. This will update the cost table to include both initial costs in the sizing calculation while ignoring the recurring costs.

The Sizing Objective window should now appear as shown in Figure 2.

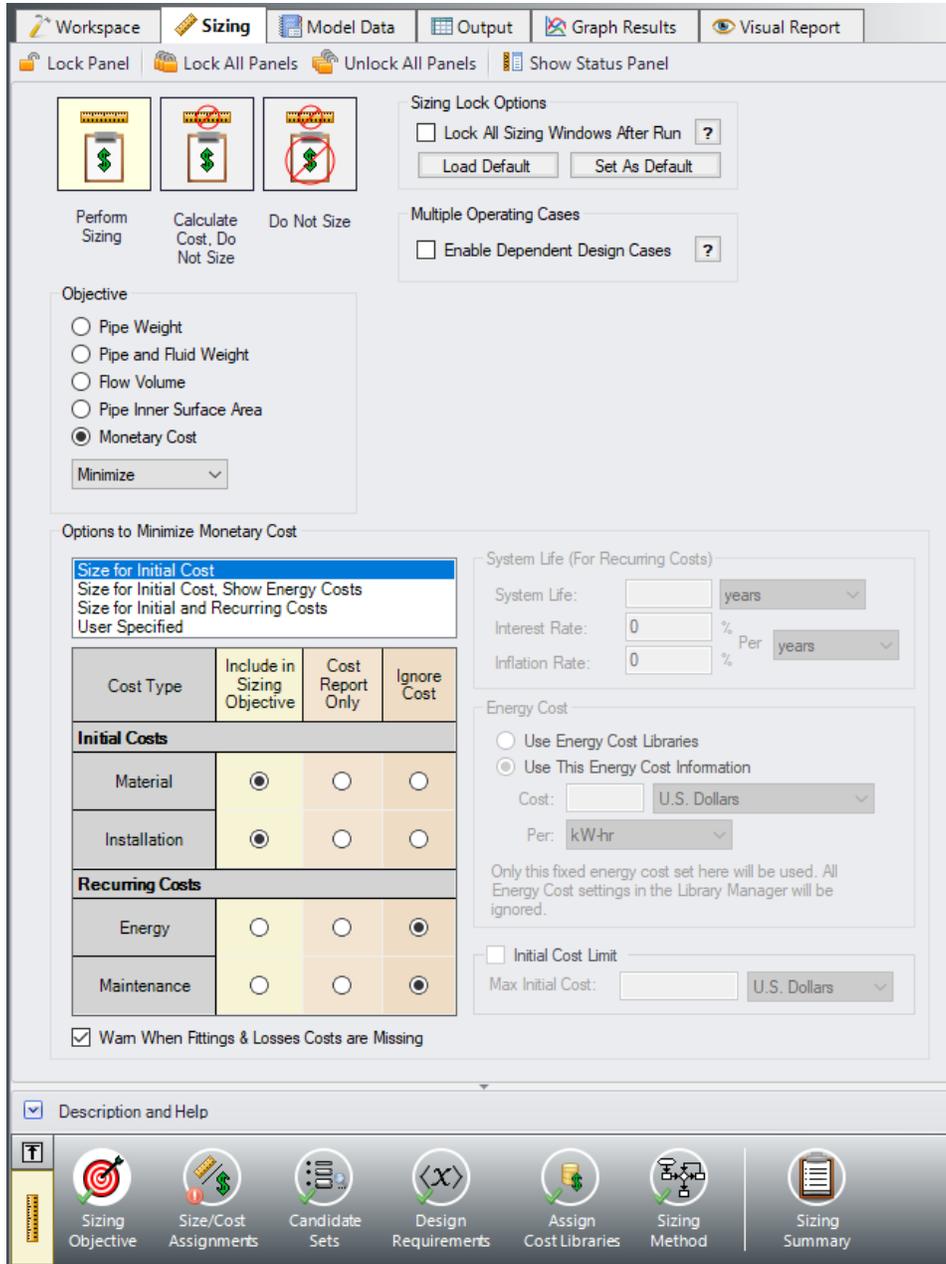


Figure 2: Sizing Objective window setup for an Initial Cost analysis

B. Size/Cost Assignment

On the toolbar at the bottom of the window select the **Size/Cost Assignments**. For this scenario we will size the neighborhood mains, system supply line and hydrant supply lines assuming that they are all the same size. The supply lines to the houses must be 1" PVC so they will not be sized, but we will still calculate the costs, since we will need to purchase that piping as well.

1. Go to the Workspace, and select the lines that will be sized as shown in Figure 3.
2. Right click the Workspace and choose **Add Pipe(s) to Common Size Group**, and select **New Group**. Give the group the name *Neighborhood Mains*. Click OK on the Information window that appears.
3. Return to the Sizing window. The neighborhood mains, supply line, and the hydrant supply lines should now all be included in the new pipe group, and should be moved to Always Include in Cost as shown in Figure 4.
4. Move each of the house supply lines to **Do Not Size - Include in Cost**. All of the pipes can be moved at once by using the shift key to highlight all of the rows, then clicking the Include in Cost button.

We are not considering any of the junctions as part of the sizing, so the Size/Cost Assignments are complete for now.

Note: Though the hydrant supply lines are closed in the primary design case, we need to include them in the Common Size Group so that they will be sized when we create the dependent design cases for the open hydrant cases.

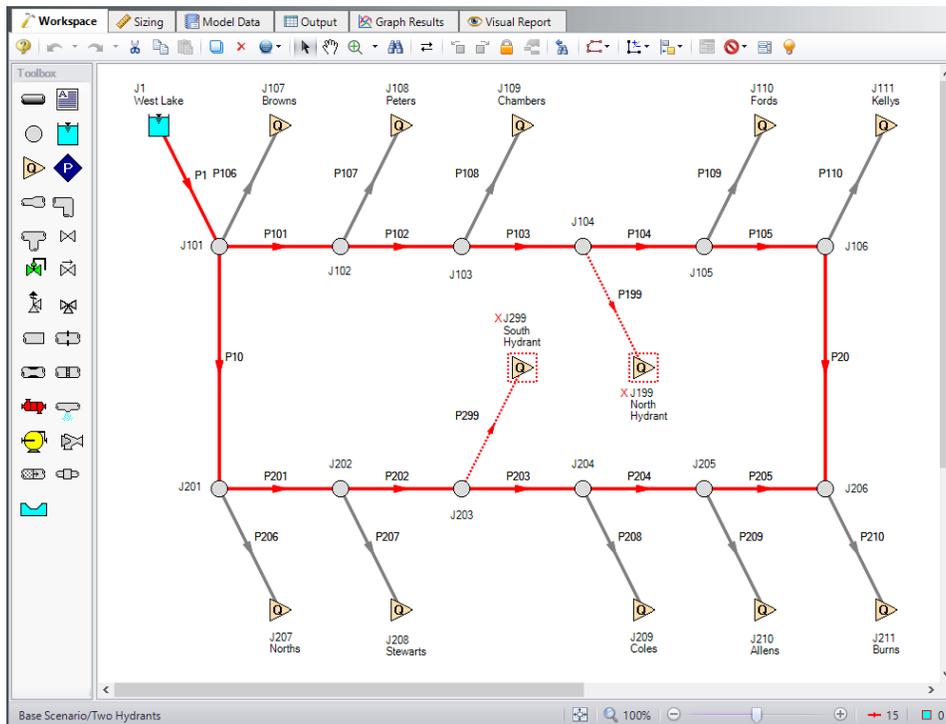


Figure 3: Workspace with pipes selected to add to the Common Size Group

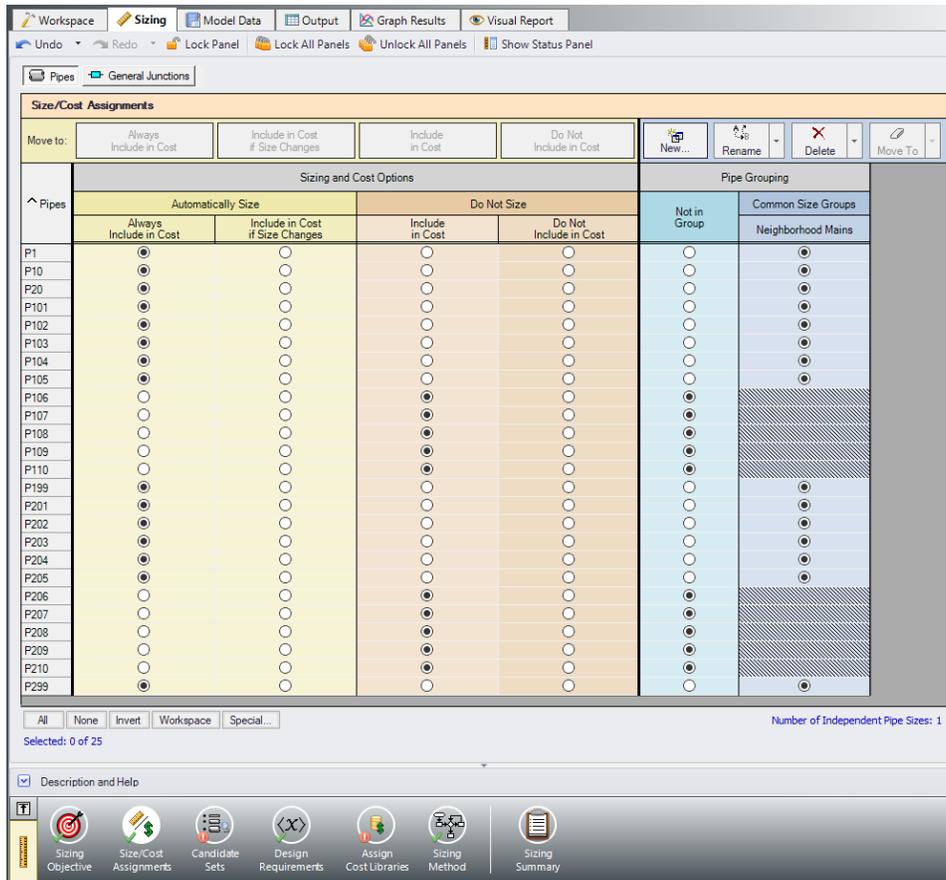


Figure 4: Size/Cost Assignments window with Common Size Groups defined as outlined above

C. Candidate Sets

Click on **Candidate Sets** to open the Candidate Sets panel. In this case we will be considering schedule 40 PVC pipe within the range of 1" to 10".

1. Select **New**, and name the candidate set *PVC sch 40*.
2. Choose **PVC - ASTM** from the material list.
3. At the bottom of the window select to sort by **Type, Schedule, Class** (if not already selected).
4. Expand the schedule 40 type, then double click each of the sizes **from 1 inch to 10 inch** to add them to the Candidate Set.
5. Click **OK** to accept the defined set.

In the bottom section of the window make sure that the Common Size Group is assigned to use the new PVC sch 40 Candidate Set. The window should appear as shown in Figure 5.

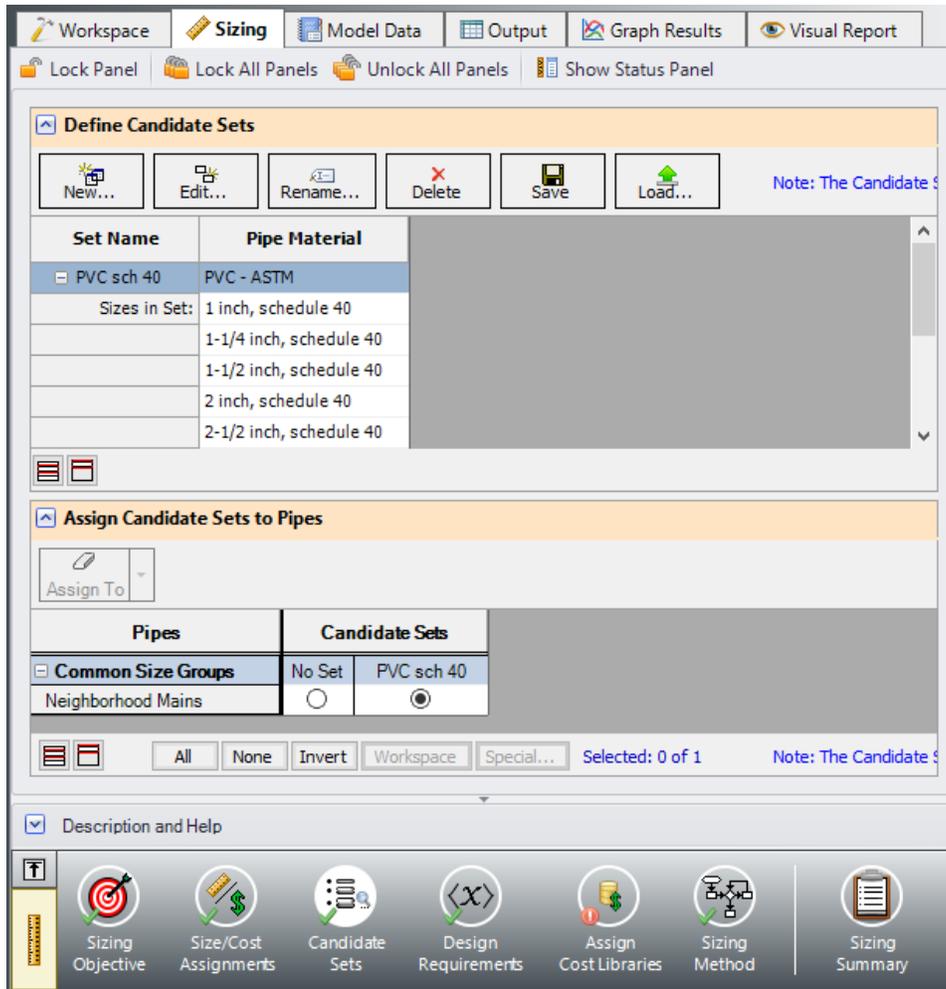


Figure 5: Candidate Sets window fully defined for the model

D. Define Design Requirements

Select Design Requirements.

For the standard operation scenario the only design requirement is for the minimum supply pressure of 4.0 barG (400 kPa(g)) to each of the houses. With the **Pipes** button selected click **New** to add a *House Supply* Design Requirement specifying the **Pressure Static** at the **Outlet** of the pipe to a **Minimum** value of **4.0 barG (400 kPa (g))**.

Click **New** again to create a second Design Requirement named *Fire Supply* specifying the **Pressure Static** at the **Outlet** of the pipe to a **Minimum** value of **6.2 barG (620 kPa (g))**. We will apply this requirement later when the dependent design cases are created.

In the bottom section of the window, make sure that the House Supply Design Requirement is applied to only the supply lines to the houses and not the Neighborhood Mains, as shown in Figure 6.

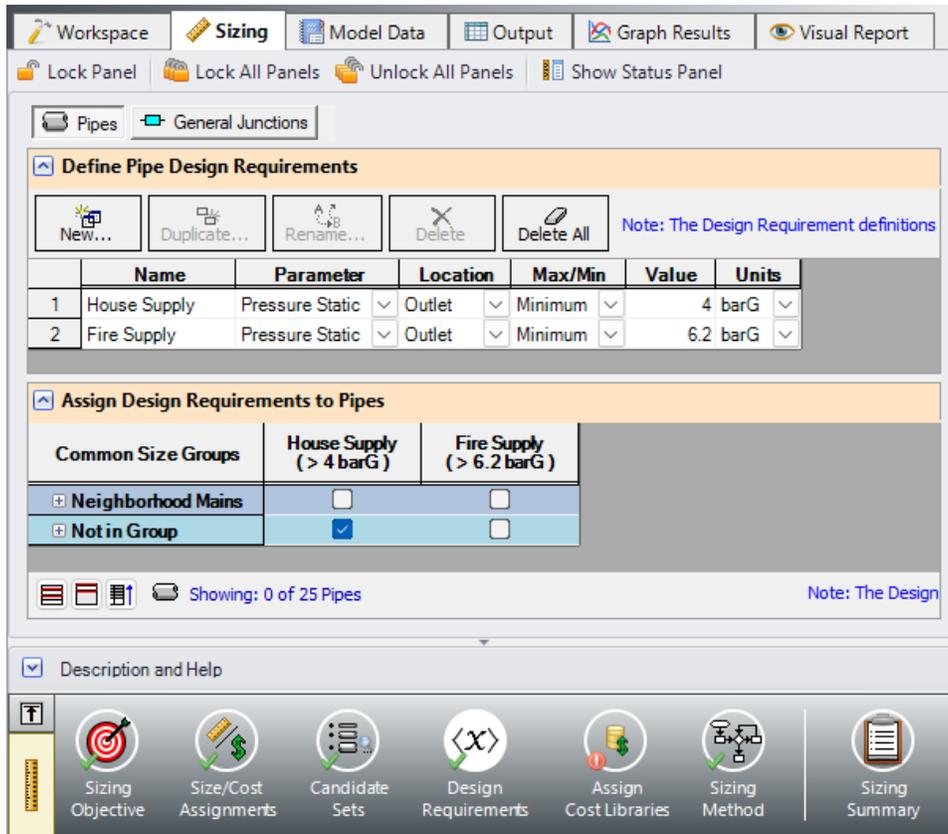


Figure 6: Design Requirements defined for the primary design case

E. Assign Cost Libraries

Select **Assign Cost Libraries**. For this model the pipe cost library has already been created, but we will need to connect and apply it.

➤ **Open the Library Manager** by clicking the Library Manager button to see which libraries are connected.

This example uses a pre-built cost library for the pipes containing the PVC material costs. To connect the library do the following:

1. Click **Add Existing Cost Library**.
2. Browse to the Fathom 13 Examples folder (located by default in C:\AFT Products\AFT Fathom 13\Examples), and open the file titled **Housing Project Costs.cst**.

The new library should now be visible in the list and be connected to the model, indicated with a box checked next to the name, as shown in Figure 7. The cost library for the pipes should appear with the name **Housing Project Costs.cst** and be indented under the PVC - ASTM library within the standard pipe material library list.

Once you have confirmed that the library is connected, click Close to exit the Library Manager.

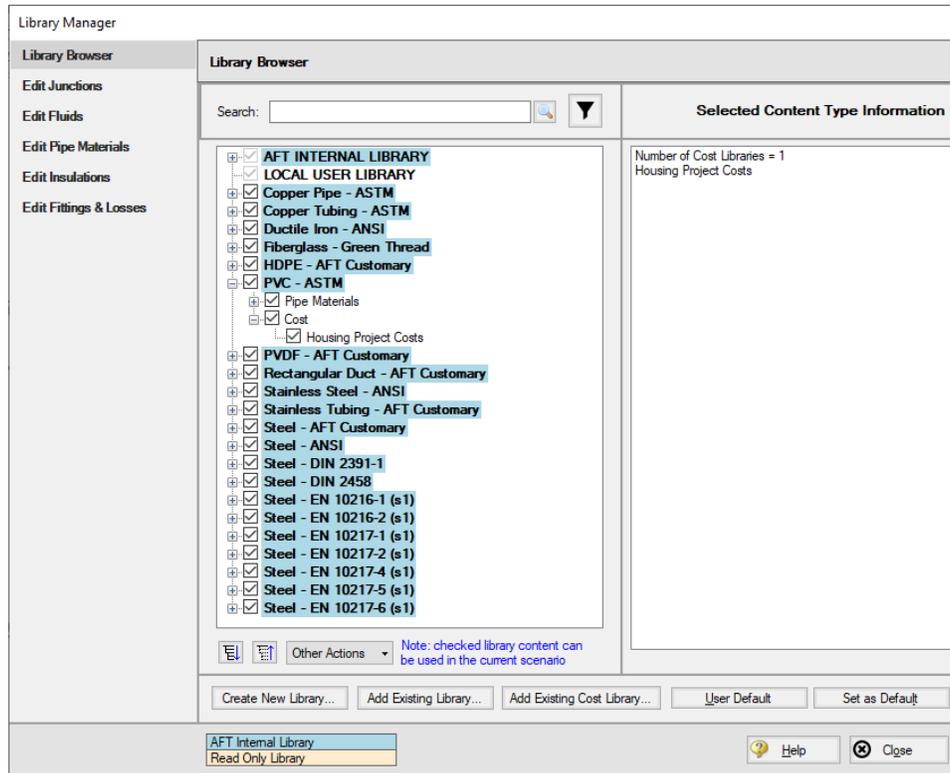


Figure 7: Library Manager with Housing Project Costs library connected

Back in the Assign Cost Libraries panel for the Pipes, the Housing Project Costs library should be the only available cost library, and should be selected to show that it is being applied to the candidate set and to each of the house supply lines.

F. Sizing Method

Select **Sizing Method** to go to the Sizing Method panel.

Ensure that **Discrete Sizing** is chosen with the **Modified Method of Feasible Directions (MMFD)** for the Search Method.

G. Dependent Design Cases

We have now set up the system to be sized for the primary design case, but we will also need to account for the cases where either the north or south hydrant are in use. There are a few changes that occur for each of these cases where a hydrant is in use:

1. The flow rate at each of the houses decreases to 0.5 m³/hr
2. The minimum inlet pressure for each of the hydrants must be 6.2 barG (620 kPa(g))
3. The minimum pressure for each of the houses is removed

We will create two dependent design cases to account for the scenarios where the North Hydrant or the South Hydrant is open. To make the dependent design cases complete the following:

1. Go to the **Sizing Objective** panel and click the box next to **Enable Dependent Design Case**. A new button will now be available for the Dependent Design Cases panel in the Sizing Navigation panel.
2. Navigate to the Dependent Design Case panel. You should now see instructions displayed to create Dependent Design Cases, along with a summary table of dependent design settings. We will now need to use the Duplicate Special feature to create the dependent design cases.
3. Go to the **Workspace** and go to the Edit menu and choose **Select All**.
4. Go to the Edit menu again and select **Duplicate Special**, enter an increment of **400** and click the boxes next to **Make Dependent Design Case** and **Clear Design Requirements** (Figure 8). Click **OK**.
5. Move the duplicated pipes and junctions below the original ones on the Workspace to distinguish them from the Primary Design Case. Click **Paste**.
6. Select only the primary design case model and repeat steps 4 and 5 to create a second dependent design case (increment the pipe numbers by **1000** to avoid conflicting pipe numbers). There should now be two copies of the original model in the Workspace which will be used to represent the alternate operating cases.
7. Open the **J599** and **J1299** hydrants by selecting them and toggling the **Special Condition to None** so that the North Hydrant is open in the second case and the South Hydrant is open in the third case.
8. Use **Global Junction Edit** to change all 20 of the Assigned Flow junctions at the Dependent Design Case homes (junctions 500's, 600's, 1100's, and 1200's) to an Outflow of **0.5 m³/hr** (see Figure 9). Make sure to not change the Assigned Flow junctions that represent the hydrants.
9. In the Sizing window navigate to the **Design Requirements** panel. Assign the **Fire Supply** Design Requirement created earlier to pipe **P599** and **P1299** by checking the adjacent boxes.

Return to the **Dependent Design Cases** panel.

When Duplicate Special was performed with Dependent Design Case (DDC) selected, each of the duplicated pipes was created with a special type of grouping. For example, pipes 401 and 1001 were grouped with pipe 1 as DDC pipes (see Figure 10). This type of assignment allows the dependent design pipes to be sized, but to not be counted in the cost so that the cost will not be duplicated.

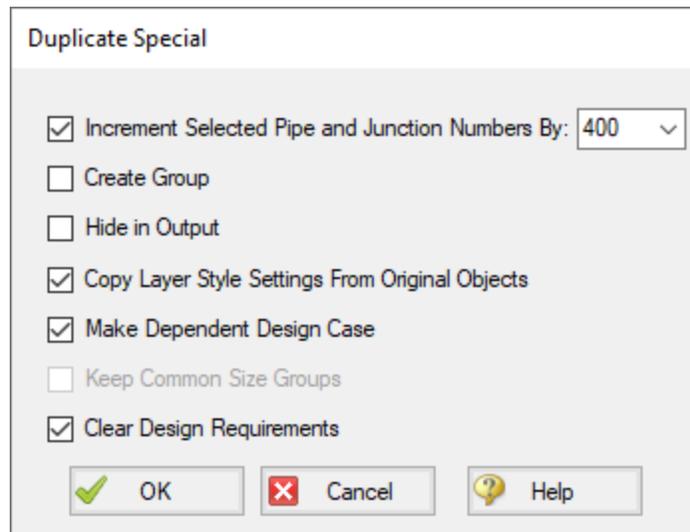


Figure 8: Duplicate Special settings to create a dependent design case

Housing Project - ANS

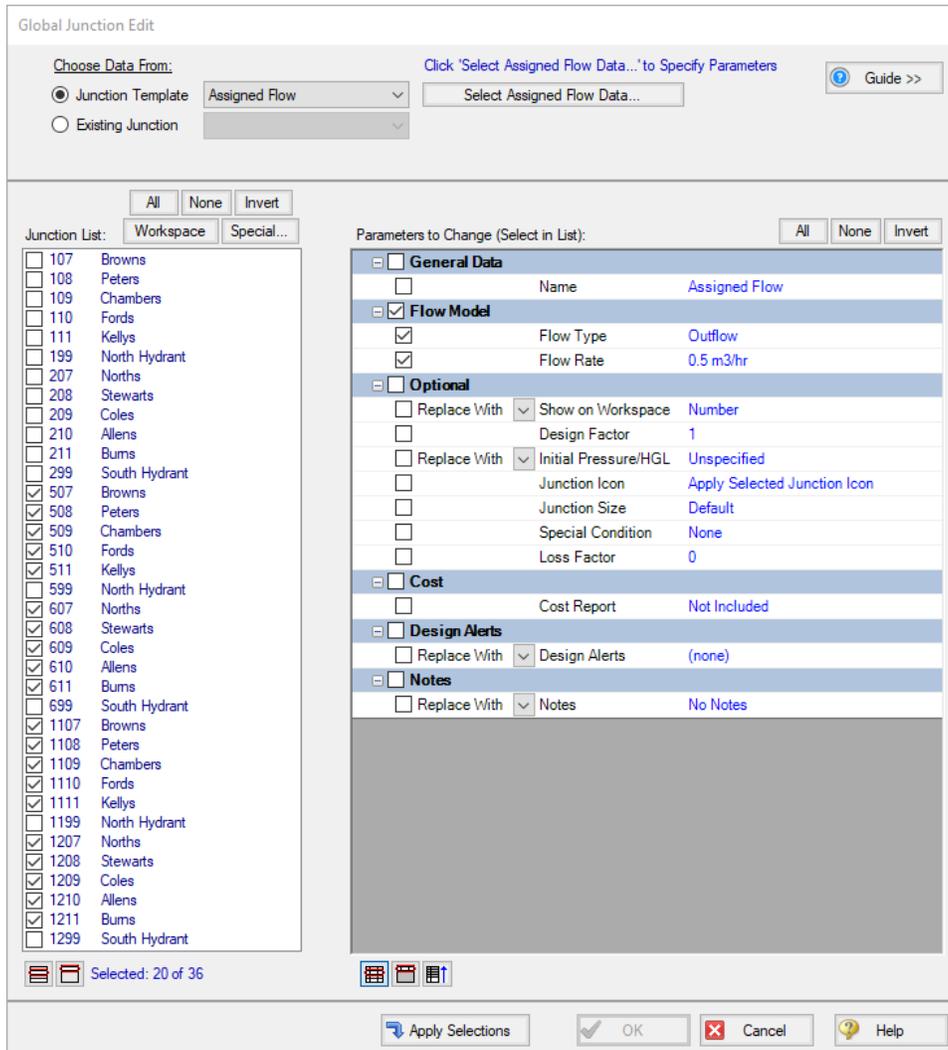


Figure 9: Global Junction Edit is used to change the flows in the homes for the dependent cases

The screenshot shows a software interface with a toolbar at the top containing buttons for Workspace, Sizing, Model Data, Output, Graph Results, and Visual Report. Below the toolbar are options for Lock Panel, Lock All Panels, Unlock All Panels, and Show Status Panel. The main area is titled 'Pipes' and 'General Junctions'. A section titled 'Dependent Design Cases' contains a table with the following data:

Pipes	DDC Pipe?	Assigned Design Requirements
P1	NO	None
P401	YES (P1)	None
P1001	YES (P1)	None
P10	NO	None
P410	YES (P10)	None
P1010	YES (P10)	None
P20	NO	None
P420	YES (P20)	None
P1020	YES (P20)	None
P101	NO	None
P501	YES (P101)	None
P1101	YES (P101)	None
P102	NO	None
P502	YES (P102)	None
P1102	YES (P102)	None
P103	NO	None
P503	YES (P103)	None
P1103	YES (P103)	None
P104	NO	None
P504	YES (P104)	None
P1104	YES (P104)	None

Below the table is a 'Description and Help' section with a checked box. At the bottom is a toolbar with icons for Sizing Objective, Size/Cost Assignments, Candidate Sets, Design Requirements, Assign Cost Libraries, Sizing Method, Dependent Design Cases, and Sizing Summary.

Figure 10: Pipes in Dependent Design Cases have a special grouping relationship with pipes in the primary case

Step 5. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Step 6. Review the Sized Results

After the run finishes, examine the final pipe sizes calculated by the ANS module. The results for pipe size and overall cost are shown in Figure 11. One can see that the resultant size is 4-inch pipe. The lines to the houses are listed as N/A on the sizing output since they were not included in the sizing.

The cost for all sized pipes is \$63,892. This is the cost for Items in Sizing. Since we did not size the pipes to the homes (because these were fixed at 1- inch), there is an additional cost of \$7,161 for Items Not In Sizing. The pipes that were sized will be distinguished in the Cost Report by having a green background color. The total cost is merely the sum of the two: \$71,053.

It should be noted that if we did not have monetary cost information for the pipes in this example, we could get nearly identical sizing results for this system by choosing a non-monetary objective. A monetary cost objective is primarily necessary in order to size equipment like pumps, control valves, etc. with the pipes.

Housing Project - ANS

⤴ General Warnings Design Alerts Cost Report Reservoir Summary					
Table Units: U.S. Dollars		Material	Installation	TOTAL	
TOTAL OF ALL MODEL COSTS				71,053	
Total of All Shown Costs		24,545	46,508	71,053	
Items In Sizing		22,671	41,220	63,892	
Items Not In Sizing		1,873	5,288	7,161	
Pipe Subtotal		24,545	46,508	71,053	
o	P1	9,906	18,012	27,918	
o	P10	1,462	2,657	4,119	

⤴ Pipes Sizing Pipe Design Requirements Library Sources					
Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (cm)
1	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
10	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
20	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
101	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
102	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
103	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
104	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
105	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
106	N/A	N/A	N/A	N/A	N/A
107	N/A	N/A	N/A	N/A	N/A
108	N/A	N/A	N/A	N/A	N/A
109	N/A	N/A	N/A	N/A	N/A
110	N/A	N/A	N/A	N/A	N/A
199	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
201	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
202	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
203	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
204	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
205	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
206	N/A	N/A	N/A	N/A	N/A
207	N/A	N/A	N/A	N/A	N/A
208	N/A	N/A	N/A	N/A	N/A
209	N/A	N/A	N/A	N/A	N/A
210	N/A	N/A	N/A	N/A	N/A
299	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
401	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
410	Pipe	PVC - ASTM	4 inch	schedule 40	10.226

Figure 11: Cost summary and final pipe sizes

Step 7. Increase the Independent Pipe Sizes

To increase savings in the system we can remove some of the pipes from the Common Size Group, which will allow them to be varied separately and increase the number of Independent Pipe Sizes for the model.

Let's create a child scenario to see how much savings can be found.

In the Scenario Manager, right-click the Two Hydrants scenario, click **Create Child**, and name it *3 Pipe Sizes*. Navigate to the **Size/Cost Assignments** panel in the Sizing window. Unlock the panel if it has been locked and move pipe **P1** to **Not in Group**. Lastly, create a new Common Size Group by clicking **New**, naming it *Hydrant Pipes*, and moving pipes **P199** and **P299** to the new group.

Run the scenario and go to the Output to view the results.

The Cost Report and pipe size results for the 3 Pipe Sizes scenario shown in Figure 12. The total cost is the cost per unit length for material and installation multiplied by the length of each section. The cost data does not include excavation costs. Even with these rough costs, significant savings can be achieved. If the developer is willing to use three pipe sizes, the savings will be approximately \$2,500 or 3.5% of the total cost.

With this example, we can see that the limitations placed on the system by Common Size Groups can impact the sizing results.

In theory, to obtain the best possible sizing results, one can ungroup all the pipes in the neighborhood and allow each pipe to be sized independently. However, in practice this can result in increased fittings, installation, and maintenance costs, which is why common size groups are desirable.

Housing Project - ANS

^ General Warnings Design Alerts Cost Report Reservoir Summary					
Table Units:		Material	Installation	TOTAL	
U.S. Dollars					
TOTAL OF ALL MODEL COSTS				68,603	
Total of All Shown Costs		22,997	45,606	68,603	
Items In Sizing		21,124	40,318	61,442	
Items Not In Sizing		1,873	5,288	7,161	
Pipe Subtotal		22,997	45,606	68,603	
o	P1	9,906	18,012	27,918	

^ Pipes Sizing Pipe Design Requirements Library Sources					
Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (cm)
1	Pipe	PVC - ASTM	4 inch	schedule 40	10.226
10	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	9.012
20	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	9.012
101	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	9.012
102	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	9.012
103	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	9.012
104	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	9.012
105	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	9.012
106	N/A	N/A	N/A	N/A	N/A
107	N/A	N/A	N/A	N/A	N/A
108	N/A	N/A	N/A	N/A	N/A
109	N/A	N/A	N/A	N/A	N/A
110	N/A	N/A	N/A	N/A	N/A
199	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	9.012
201	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	9.012
202	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	9.012
203	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	9.012
204	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	9.012
205	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	9.012
206	N/A	N/A	N/A	N/A	N/A
207	N/A	N/A	N/A	N/A	N/A
208	N/A	N/A	N/A	N/A	N/A
209	N/A	N/A	N/A	N/A	N/A
210	N/A	N/A	N/A	N/A	N/A
299	Pipe	PVC - ASTM	3-1/2 inch	schedule 40	9.012

Figure 12: Cost Report and pipe sizes for the system with three independently varied pipe sizes

Combined Modules Examples

The examples below utilize multiple AFT Fathom modules.

Combined Module Examples

Example	Complexity	Fluid	Pipes	Pumps	Description
Fixed Head Supply Tank	Advanced	Water	3	1	Learn how to use the GSC module, XTS module, and cost calculations together to model system performance over time.

Fixed Head Supply Tank - GSC XTS

Summary

Previous examples have illustrated how to use each of the AFT Fathom modules by themselves. However, AFT Fathom also allows you to use the modules together, in any combination you choose. This allows even greater flexibility in the types of analyses that can be accomplished using AFT Fathom.

This example shows how the GSC and XTS modules and cost calculations can be performed simultaneously by examining a fixed head supply tank system.

A process plant requires the delivery of water at a fixed head, regardless of the system demand. This is accomplished by pumping the water to a supply reservoir, and maintaining a constant fill-level by varying the supply pump speed as the process demands change. Throughout any 24-hour period the process demands vary from as little as 115 m³/hr to peak demands of 570 m³/hr.

Use the GSC and XTS modules to analyze the system performance over a 24-hour period. Perform cost calculations to determine the energy costs for the pump with this daily operation over one year.

Note: This example can only be run if you have licenses for the GSC and XTS modules.

Topics Covered

- Using the GSC and XTS modules and performing cost calculations simultaneously

Required Knowledge

This example assumes the user has already worked through the [Beginner - Three Reservoir](#) example, or has a level of knowledge consistent with that topic. You can also watch the [AFT Fathom Quick Start Video Tutorial Series](#) on the AFT website, as it covers the majority of the topics discussed in the Three-Reservoir Problem example.

In addition, it is assumed that the user has worked through the [Beginner -Filling a Tank - XTS](#) example, the [Beginner - Heat Transfer in a Pipe - GSC](#) example, and the [Beginner - Cost Calculation](#) example and is familiar with the basics of GSC, XTS module, and cost calculation analyses.

Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Fathom installation:

- *Metric - Fixed Head Supply Tank - GSC XTS.fth*

Step 1. Start AFT Fathom

From the Start Menu choose the AFT Fathom 13 folder and select AFT Fathom 13.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Fathom settings, unless you are specifically instructed to do otherwise.

Step 2. Define the Modules Panel

Open Analysis Setup from the toolbar or from the Analysis menu. Navigate to the Modules panel. For this example, check the box next to Activate GSC and Activate XTS. Make sure Use is selected for GSC and Transient is selected for XTS. Two new groups will appear in Analysis Setup titled Goal Seek and Control and Transient Control. Click OK to save the changes and exit Analysis Setup. Open the Analysis menu to see the two new options called Goal Seek & Control and Time Simulation. From here you can quickly toggle between Ignore (normal AFT Fathom) and Use (GSC mode) for GSC and between Steady Only (normal AFT Fathom) and Transient (XTS mode).

Step 3. Define the Fluid Properties Group

1. Open Analysis Setup from the toolbar or from the Analysis menu.
2. Open the Fluid panel then define the fluid:
 - a. Fluid Library = AFT Standard
 - b. Fluid = Water (liquid)
 - i. After selecting, click Add to Model
 - c. Temperature = 21 deg. C

Step 4. Define the Pipes and Junctions Group

At this point, the first two groups are completed in Analysis Setup. The next undefined group is the Pipes and Junctions group. To define this group, the model needs to be assembled with all pipes and junctions fully defined. Click OK to save and exit Analysis Setup then assemble the model on the workspace as shown in the figure below.

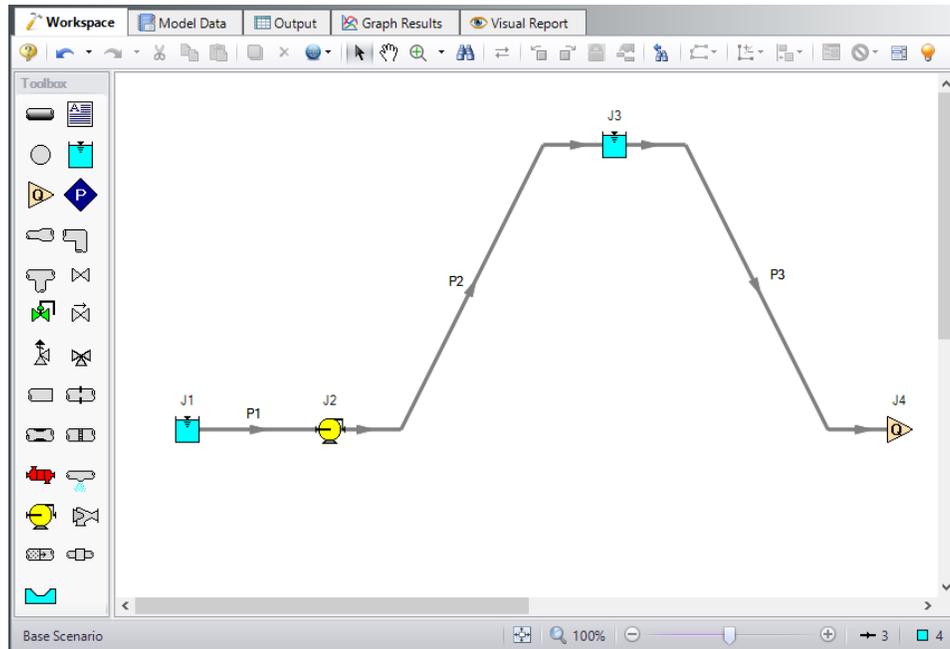


Figure 1: Fixed Head Supply Tank model layout

Pipe Properties

1. Pipe Model tab
 - a. Pipe Material = Steel - ANSI
 - b. Pipe Geometry = Cylindrical Pipe
 - c. Size = 5 inch
 - d. Type = STD (schedule 40)
 - e. Friction Model Data Set = Standard
 - f. Lengths =

Pipe	Length (meters)
1	9
2	150
3	30

Junction Properties

1. J1 Reservoir

- a. Tank Model = Infinite Reservoir
- b. Liquid Surface Elevation = 9 meters
- c. Liquid Surface Pressure = 0 barG
- d. Pipe Depth = 9 meters

2. J2 Pump

- a. Inlet Elevation = 0 meters
- b. Pump Model = Centrifugal (Rotodynamic)
- c. Analysis Type = Pump Curve
- d. Enter Curve Data =

Volumetric	Head	Efficiency
m3/hr	meters	Percent
0	150	0
280		60
570	145	80
850		70
1135	75	40

- e. Curve Fit Order = 2

3. J3 Reservoir

- a. Name = Supply Tank
- b. Tank Model = Finite Open Tank
- c. Known Parameters Initially = Both Parameters (Liquid Surface Level & Surface Pressure)
- d. Cross-Sectional Area = Constant, 29 meters²
- e. Liquid Level Specification = Height from Bottom
- f. Liquid Height = 5 meters
- g. Liquid Surface Pressure = 0 barG
- h. Tank Height = 6 meters
- i. Pipe Depth = 5 meters
- j. Tank Bottom Elevation = 6 meters

3. J4 Assigned Flow

- 1. Elevation = 0 meters
- 2. Type = Outflow
- 3. Flow Specification = Volumetric Flow Rate
- 4. Flow Rate = 115 m3/hr

Step 5. Define the Goal Seek and Control Group

Specify the variables and goals for the model.

Variables Panel

Open the Variables panel in the Goal Seek and Control group in Analysis Setup. For this example, we will be adding one pump variable: pump speed.

Click **New Variable** and change the Junction Type to **Pump**. By default, J2 (Pump) and Variable Speed will be selected for Object Number and Name and Variable Parameter, respectively. The Variables panel should look now like Figure 2.

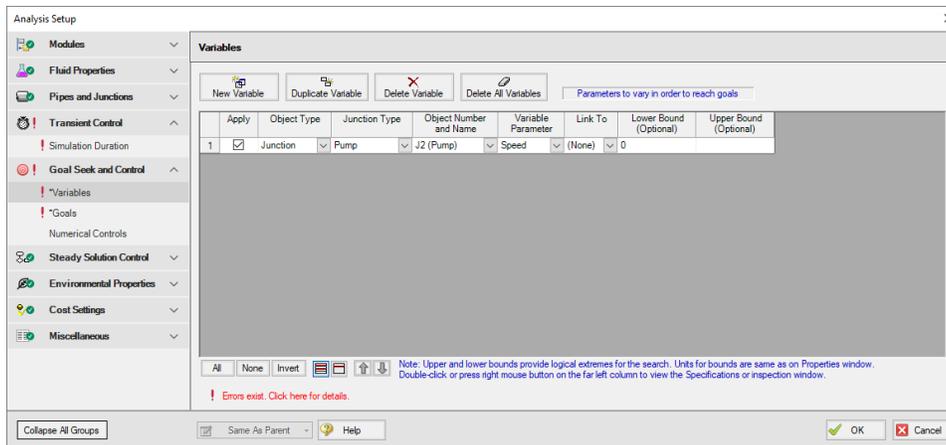


Figure 2: Defined Variables panel

Goals Panel

Navigate to the Goals panel. The desired goal is for the net volumetric flow rate into the junction to be zero, which will keep the water level in the tank constant.

Click **New Goal** and change the Object Type or **Reservoir**. By default, Vol. Flow Rate New Into Jct should be the selected Goal Parameter. Enter **0** as the Goal Value and select **J3 (Supply Tank)** for the Object ID. The Goals panel should now look like Figure 3.

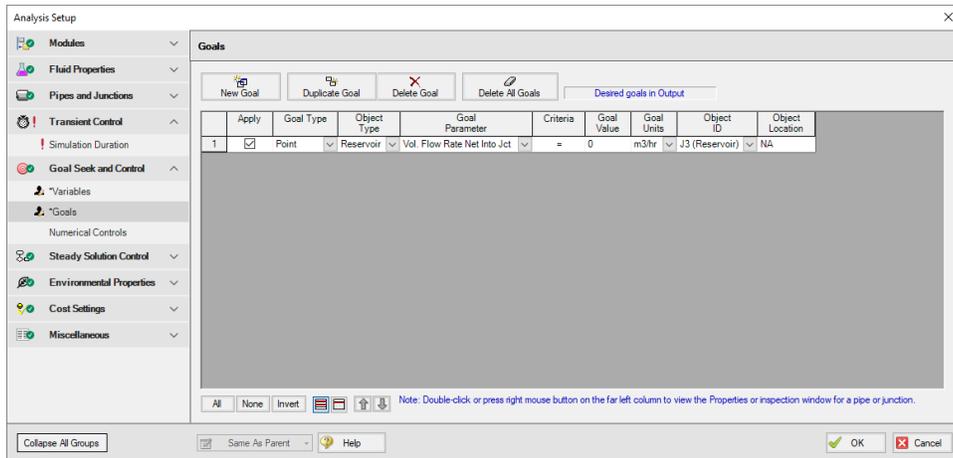


Figure 3: Defined Goals panel

Step 6. Define the Transient Control Group

The XTS module is used to model the process demand changes over the 24-hour analysis period. Navigate to the Simulation Duration panel in the Transient Control group to view the Transient Control settings. Change the Start Time units to **hours** and enter a Stop Time of **24**. For Time Step, changes the units to **minutes** and enter **10**. The Simulation Duration panel should now look like Figure 4. The transient analysis is now set to run for a period of 24-hours with a time step of 10 minutes. Click **OK** to save and close Analysis Setup.

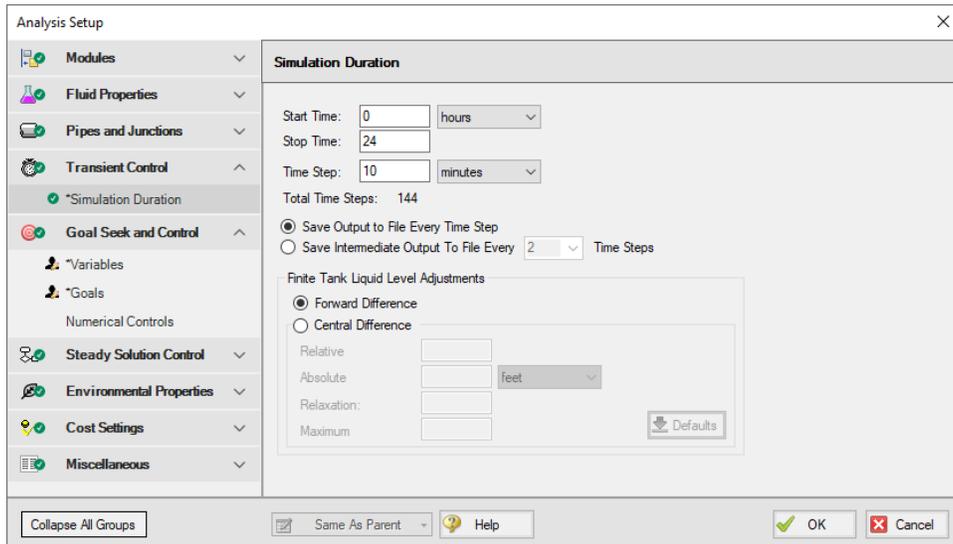


Figure 4: Transient Control settings for a 24-hour period

Step 7. Junction Transient Data

The flow variation in this model is driven by the J4 Assigned Flow junction. Open the J4 Assigned Flow Properties window and select the Transient tab. The process flow demands are modeled as time-varying flows. The flow profile over the 24-hour period is defined in the Transient Data table. Use Table 1 below to fill in the Transient Data table. Then, click Show Graph and make sure the Junction J4 Transient Data graph matches Figure 5.

Note that the time units are in seconds. Alternatively, you can change the preferred time units in User Options from seconds to hours to change the units of this table, then enter the values in this table in hours instead. However, to keep to the default units, this table is provided in time units of seconds.

Table 1: Transient Data table for J4 Assigned Flow

Data Point	Time (seconds)	Vol. Flow (m3/hr)
1	0	115
2	7200	115
3	14400	340
4	25200	450
5	28800	450
6	32400	570
7	43200	230
8	50400	115
9	57600	115
10	64800	340
11	68400	450
12	75600	450
13	82800	230
14	86400	115

Combined Modules Examples

Assigned Flow Properties

Number: Upstream Pipe:
Name: Downstream Pipe:
Library Jct:
Copy Data From Jct...:

Elevation:

Flow Model Transient Optional Design Alerts Notes Status

Transient Special Condition
 None
 Ignore Transient Data

Initiation of Transient
 Time Dual Event Cyclic
 Single Event Dual Event Sequential

Transient Data
 Absolute Values
 Relative To Steady-State Value

Data Point	Time (seconds)	Vol. Flow (m3/hr)
1	0	115
2	7200	115
3	14400	340
4	25200	450
5	28800	450
6	32400	570
7	43200	230
8	50400	115
9	57600	115
10	64800	340
11	68400	450
12	75600	450

Repeat Transient

Transient

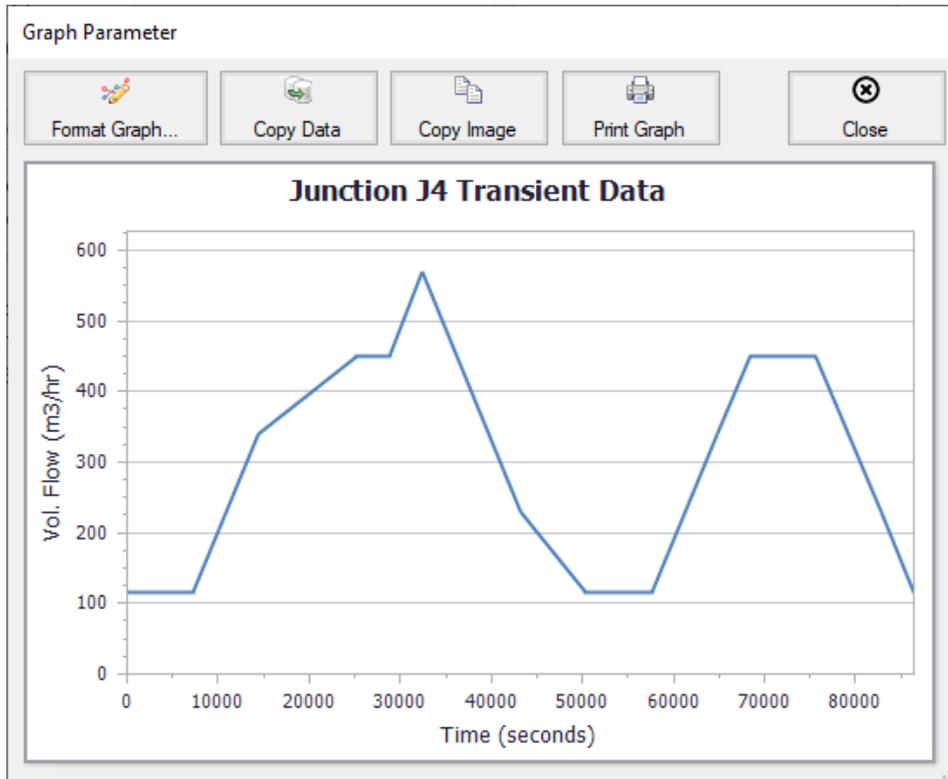


Figure 5: Flow demand over 24 hours at the J4 junction

In addition, the J3 Reservoir junction is modeled as a finite reservoir (Figure 6). The J3 Reservoir can potentially drain and fill, and the varying pump speed (and, hence, flow) is what will keep its liquid level at 15 feet. No further input is required for the Supply Tank.

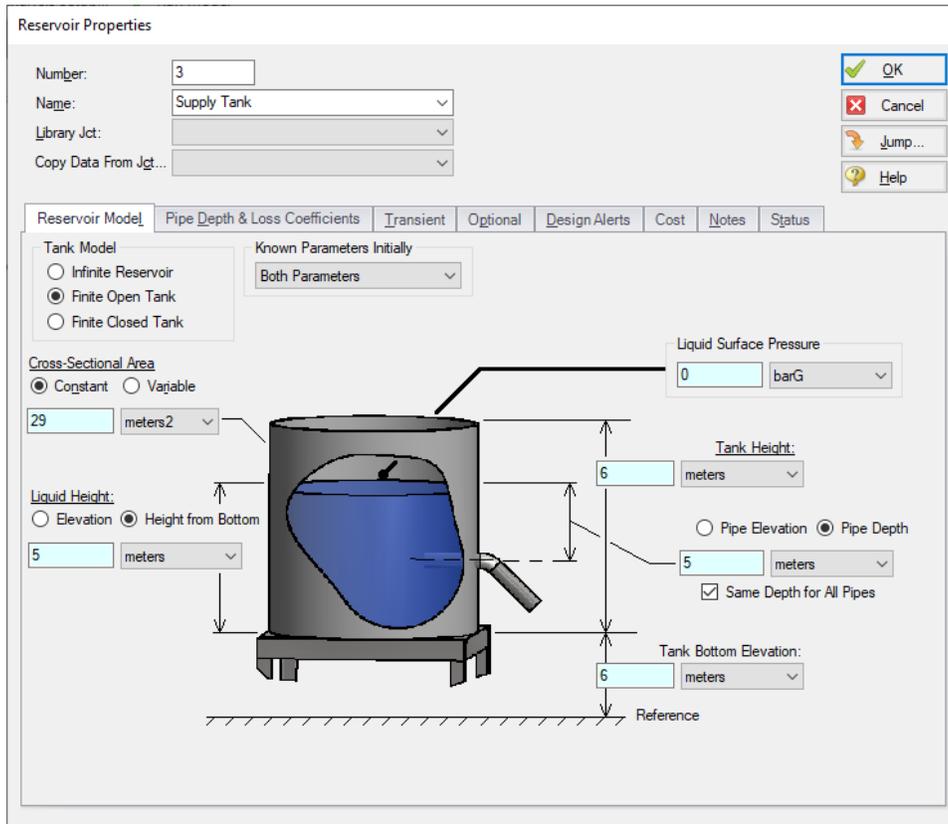


Figure 6: J3 Reservoir junction is finite, which means it can drain and fill

Step 8. Cost Settings

For this problem, the pump energy cost for the 24-hour analysis period is entered as a fixed energy cost on the Cost Settings panel. Open the Cost Settings panel from the Analysis Setup window to view the energy cost settings.

For Cost Calculation, select **Calculate**. Under Cost Definitions, check the box for **Operation/Energy** because that is the only cost of interest for this example.

Set the Cost Time Period System Life to **1 year**. The 24-hour operation is assumed to repeat every day for this one-year period, and the total energy cost will be calculated assuming the 24-hour operation repeats non-stop throughout the year. More complicated cost variation is possible when using energy cost libraries.

For Energy Cost, make sure **Use This Energy Cost Information** is selected and enter a Cost of **0.12 U.S. Dollars Per kW-hr**.

The Cost Setting panel should now look like Figure 7 below.

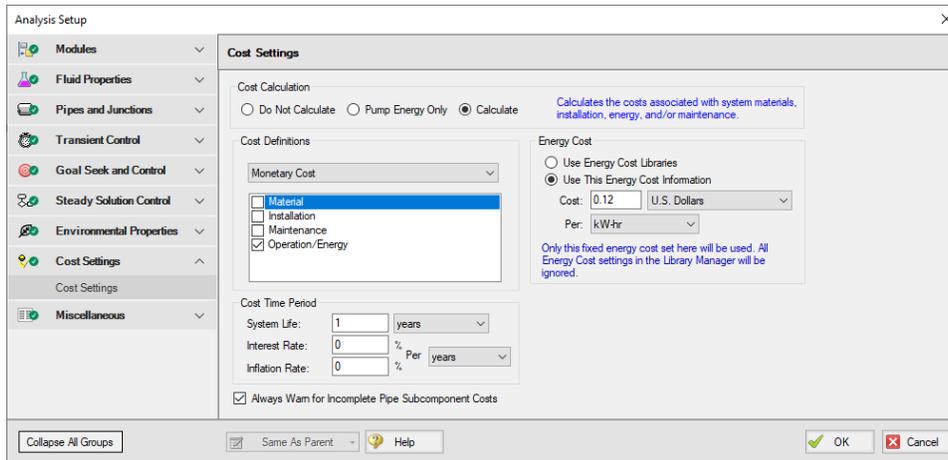


Figure 7: Cost Settings window applies Operation/Energy cost, can set the energy cost, and specifies the cost time period

Return to the Workspace and open the properties window for **Pump J2**. Navigate to the **Cost** tab and under Cost Report, select **Include Cost in Report**. Click **OK**.

Step 9. Run the Model

Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Fathom solver converges on the answer. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Note: When the GSC and XTS modules are used simultaneously, the GSC module is rerun for each time step. The GSC module variables are recalculated for the system conditions at each time step to ensure the specified goals are always met. This will typically result in longer run times, due to the increase in solution iterations that must be solved for each time step analysis.

Step 10. Examine the Output

When using multiple modules, the goal seek, transient and cost analysis results are shown in the Output window in the same manner as when they are used individually.

First open Output Control and go to the Format & Action tab. In the Time Simulation Formatting section, change the units to hours. Now all time units for XTS related output parameters will be shown using time units of hours.

The GSC variable and goal results in the General Output section can be viewed for each time step by using the slider bar at the bottom of the Output window, as shown in Figure 8.

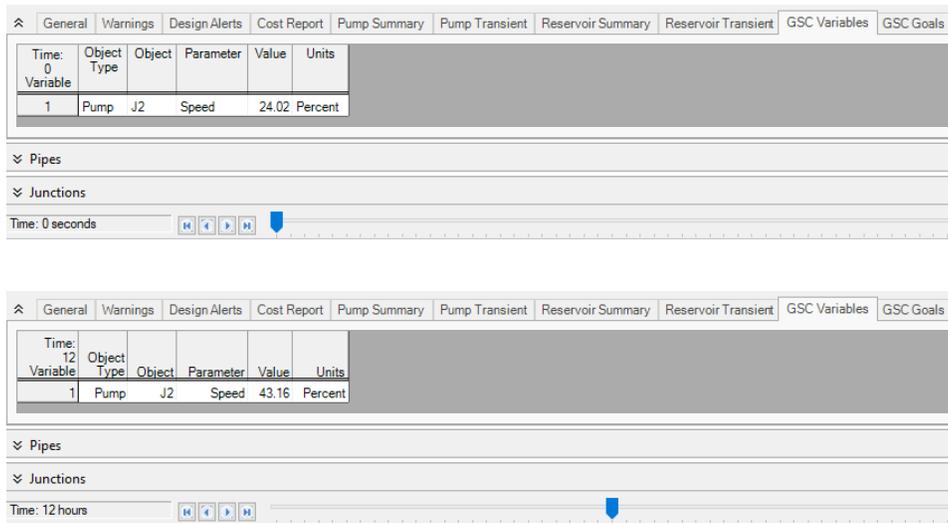


Figure 8: The slider bar at the bottom of the Output window can be used to view the transient results at any time step, including the GSC module variable and goal results. The GSC variable results for Time = 0 hours and Time = 12 hours are shown

Cost data is displayed in the Cost Report in the General Output section, as shown in Figure 9. The one-year energy cost for the supply pump in this example was \$76,457.

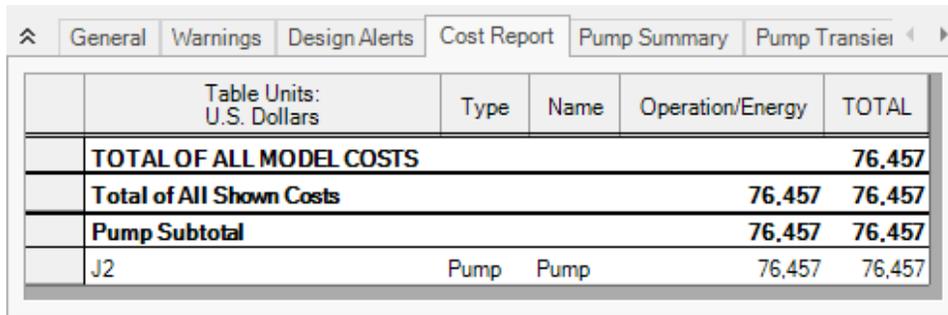


Figure 9: The cost data is shown in the Cost Report in the General Output section of the Output window

Figure 10 shows how the GSC module varied the pump's speed over time with the varying process flow rate.

Figure 11 shows that the water level in the process supply tank changes little over time because the net volumetric flow is held constant. The pump speed changes to increase and decrease the flow in order to keep the net flow into the Supply Tank zero. The minor changes in tank level are a result of the Forward Difference tank level calculation and can be reduced using the Central Difference method and a finer time step. These can be adjusted within the Transient Control section of Analysis Setup.

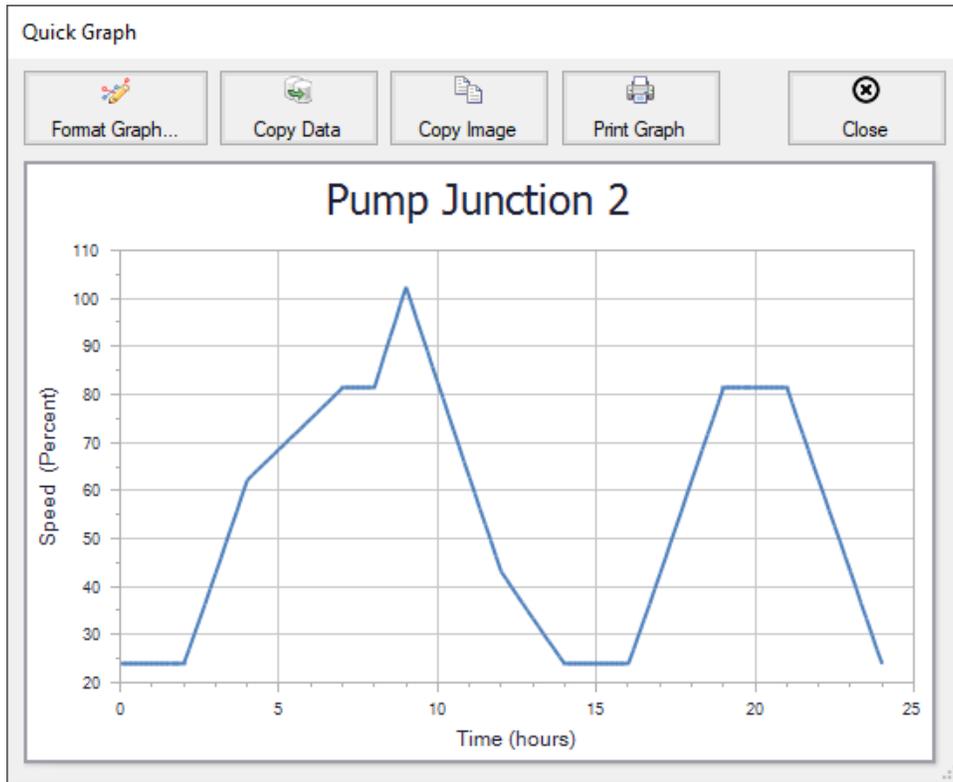


Figure 10: The GSC module varies the supply pump speed as the process flow rate changes over time (top)

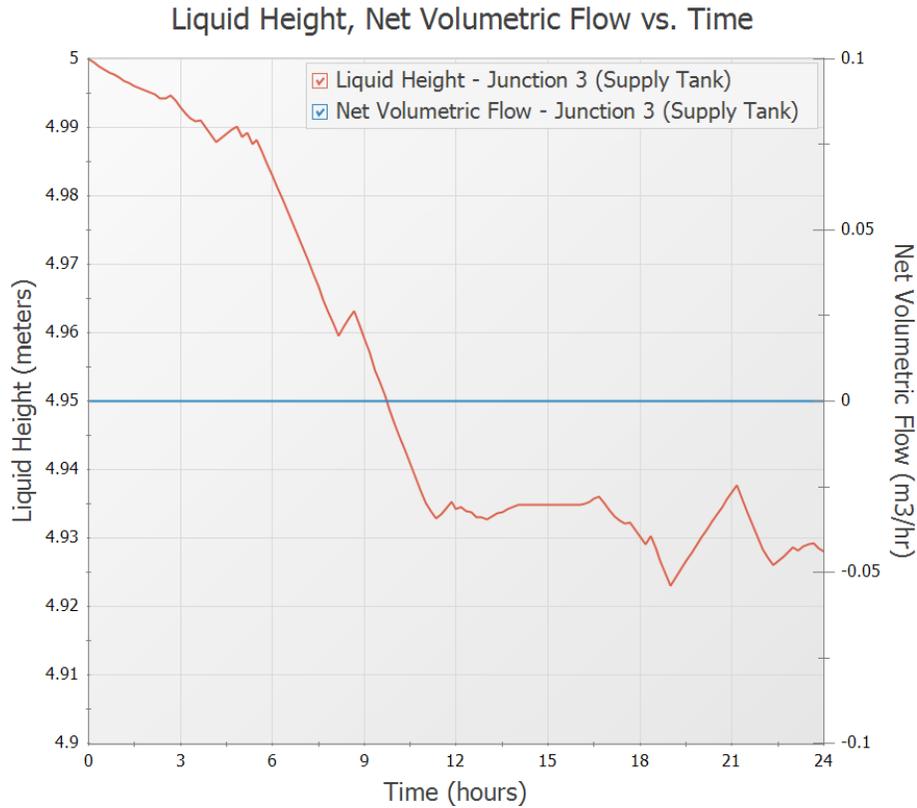


Figure 11: The water level in the process supply tank (left axis) remains relatively constant over time and the net volumetric flow rate (right axis) remains exactly at zero due to the variation in the pump speed

Summary

Individually, the AFT Fathom modules provide you with powerful analytical tools to assist you in the design of your pipe systems. The ability to combine the capabilities of these tools provides you with even greater and more powerful modeling capabilities.

Verification

AFT Fathom Verification Overview

There are a number of aspects to the verification process employed by Applied Flow Technology to ensure that AFT Fathom provides accurate solutions to incompressible pipe flow systems. These are discussed in [Verification Methodology](#). A listing of all of the verified models is given in [Summary of Verification Models](#). The verification models are taken from numerous [References](#).

References

1. [ANSI/HI Standard 9.6.7-2015](#), Rotodynamic Pumps - Guidelines for Effects of Liquid Viscosity on Performance, 2015, published by Hydraulic Institute, 6 Campus Drive, Parsippany, New Jersey, USA
2. [Crane Co.](#), Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988
3. [D.S. Miller](#), Internal Flow Systems, 2nd Ed., 1990, Gulf Publishing Co.
4. [Ernest Brater, Horace Williams, James Lindell, C.Y. Wei](#), Handbook of Hydraulics, 7th Ed., 1996, McGraw-Hill
5. [Igor Karassik, William Krutzsch, Warren Fraser, Joseph Messina](#), Pump Handbook, 2nd Ed., 1986, McGraw-Hill
6. [Ingersoll-Dresser Pumps](#), Cameron Hydraulic Data, 18th Ed., 1995, Published by Ingersoll-Dresser Pumps
7. [James John, William Haberman](#), Introduction to Fluid Mechanics, 2nd Ed., 1980, Prentice-Hall
8. [K.C. Wilson, G.R. Addie, A. Sellgren, and R. Clift](#), Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer
9. [Michael R. Lindeburg](#), P.E., Mechanical Engineering Review Manual, Seventh Edition, Professional Publications, Belmont, CA, 1984
10. [Mohinder L. Nayyar](#), Piping Handbook, Sixth Edition, McGraw-Hill, New York, 1992
11. [Nicholas P. Chopey](#), Handbook of Chemical Engineering Calculations, 1994, McGraw-Hill
12. [Robert H. Perry and Don W. Green Editors](#), Author James. N. Tilton, Perry's Chemical Engineer's Handbook, Seventh Edition
13. [Robert W. Fox and Alan T. McDonald](#), Introduction to Fluid Mechanics, Third Edition, John Wiley & Sons, 1985
14. [Roland Jeppson](#), Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science
15. [Theodore Baumeister, Eugene Avallone, Theodore Baumeister III](#), Marks' Standard Handbook for Mechanical Engineers, 8th ed., 1978, McGraw-Hill
16. [William S. Janna](#), Introduction to Fluid Mechanics, PWS Publishers, Belmont, CA 1983

Verification Methodology

Verification Methodology

The *AFT Fathom* software is an incompressible pipe flow analysis product intended to be used by trained engineers. As a technical software package, issues of quality and reliability of the technical data generated by the software are important. The following description summarizes the steps taken by Applied Flow Technology to ensure high quality in the technical data.

1. Comparisons with open literature examples

Numerous examples of pipe flow systems are available in the open literature which include published results. AFT Fathom results have been compared against many open literature systems, which include network systems up to 70 pipes in size. AFT Fathom predictions compare favorably in all cases.

2. Software checks results to ensure mass and energy balance

AFT Fathom uses a popular iterative method to obtain solutions to pipe network systems. The method is known as the Newton-Raphson method. As applied to pipe systems, the Newton-Raphson method employs the conservation of mass equation and the momentum balance equation (i.e., Bernoulli). Solutions are sought which satisfy these equations at all points in the system. After a solution is obtained, a final check is made by the software whereby the mass flow into each node is checked for balance. If a balance is not found, the user is warned in the output. This ensures that the results generated by the software agree with the applicable fundamental equations.

In addition, if heat transfer is modeled, AFT Fathom performs a final energy balance check for each junction.

See the AFT Fathom Help System for more information.

3. Software has been used in industry since April, 1994 with no significant technical errors

AFT Fathom became available in April, 1994 and is currently being used by companies in the following industries: chemical, petrochemical, power generation, architectural, ship construction, mining, automotive, aerospace, pulp and paper, pharmaceutical, municipal water, and environmental. Since its release, no significant technical errors have been found. AFT Fathom has been used to model a wide variety of systems and customers have reported good agreement with operating and/or test data where available.

In addition, Applied Flow Technology issues maintenance releases of the software periodically to improve performance and correct any problems that may have been discovered.

ANS Module Verification Methodology

Due to the nature of AFT Fathom's ANS (automated network sizing) module, there are several unique issues with verifying that the ANS module's solution represents the best configuration to minimize system cost.

The verification issues of the AFT Fathom ANS module can be summarized by two questions:

- a. ***Will the system perform as specified?*** Because the results from each possible solution are obtained from a complete run of the hydraulic engine, the final sized system will perform as specified. The accuracy of the hydraulic solution is discussed in the [Verification Methodology](#) section and is supported by 70 verification examples and years of use in industry.
- b. ***Is the sized solution the absolute optimum system?*** Because of the unique technology of ANS, this question is more difficult. The ANS module is the only automated network sizing tool available and there are no standards to compare against. We can have confidence in ANS for two reasons; first, we have verified the sizing of ANS against several literature examples of pipe and duct sizing, and, secondly, the underlying optimization technology is mature and has been used in industry for many years. If you are able to find a system that is better sized and still meets all the operational constraints of a system, please contact AFT Support.

1. Comparisons with open literature examples

To date, the ANS module has been compared with four published examples of problems that utilize the automated sizing features of the ANS module. The ANS module agrees favorably with all examples and matches exactly for discrete sizing problems where ANS is choosing from a set of nominal sizes.

2. Optimization Technology is Mature

The ANS module uses the optimization engine, VisualDOC, developed by Vanderplatts Research and Development. This optimization technology has been used by industry for many years and has been applied to a wide variety of problems in diverse engineering systems. Contact Vanderplatts Research and Development (www.vrand.com) for more information on this technology.

Verification Models

Summary of All Verification Models

Comparison of AFT Fathom predictions to the published calculation results is included herein for eighty cases from [fifteen sources](#). Below is a summary of the cases.

[Summary of Verification Models with Pumps](#)

[Summary of Verification Models with NPSH](#)

[Summary of Verification Models with PRVs](#)

[Summary of Verification Models with Slurries](#)

[Summary of Verification Models with ANS](#)

Case	Fluid	Pipes	Pumps	NPSH	PRV's	Reference
Case 1	Water	3	0	No	0	Perry's (Tilton)
Case 2	Water	11	0	No	0	Nayyar (Swierzawski)
Case 3	Water	2	1	No	0	Miller
Case 4	Air	1	0	No	0	Miller
Case 5	Water	1	0	No	0	Miller
Case 6	Water	1	0	No	0	Brater, Williams, Lindell and Wei
Case 7	Unspecified	1	0	No	0	Brater, Williams, Lindell and Wei
Case 8	Water	4	0	No	0	Brater, Williams, Lindell and Wei
Case 9	Water	3	0	No	0	Brater, Williams, Lindell and Wei
Case 10	Water	6	0	No	0	Brater, Williams, Lindell and Wei
Case 11	Water	8	1	No	0	Brater, Williams, Lindell and Wei
Case 12	Water	3	1	Yes	0	Ingersoll-Dresser Pumps
Case 13	Water	3	1	Yes	0	Ingersoll-Dresser Pumps
Case	Water	3	1	Yes	0	Ingersoll-Dresser Pumps

Summary of All Verification Models

Case	Fluid	Pipes	Pumps	NPSH	PRV's	Reference
14						
Case 15	Water	3	1	Yes	0	Ingersoll-Dresser Pumps
Case 16	Water	3	1	Yes	0	Ingersoll-Dresser Pumps
Case 17	Water	2	1	No	0	Ingersoll-Dresser Pumps
Case 18	Specific gravity = 0.8	2	1	No	0	Karassik, Krutzsch, Fraser and Messina
Case 19	Water	1	0	No	0	Karassik, Krutzsch, Fraser and Messina
Case 20	Water	1	0	No	0	Karassik, Krutzsch, Fraser and Messina
Case 21	Water	1	0	No	0	Karassik, Krutzsch, Fraser and Messina
Case 22	Oil	2	0	No	0	Karassik, Krutzsch, Fraser and Messina
Case 23	Viscous Fluid	2	1	No	0	Hydraulic Institute
Case 24	Water	1	0	No	0	Lindeburg
Case 25	Water	1	0	No	0	Lindeburg
Case 26	Water	8	1	Yes	0	Lindeburg
Case 27	Air	1	0	No	0	Lindeburg
Case 28	Water	1	0	No	0	Crane
Case 29	Water	1	0	No	0	Crane
Case 30	SAE 10 Lube Oil	1	0	No	0	Crane
Case 31	SAE 70 Lube Oil	1	0	No	0	Crane

Summary of All Verification Models

Case	Fluid	Pipes	Pumps	NPSH	PRV 's	Reference
<u>Case 32</u>	SAE 70 Lube Oil	4	0	No	0	Crane
<u>Case 33</u>	Water	10	0	No	0	Crane
<u>Case 34</u>	Fuel Oil	1	0	No	0	Crane
<u>Case 35</u>	Water	3	0	No	0	Crane
<u>Case 36</u>	Water	6	0	No	0	Crane
<u>Case 37</u>	Crude Oil 30 degree API	2	0	No	0	Crane
<u>Case 38</u>	Water	4	0	No	0	Crane
<u>Case 39</u>	Water	1	0	No	0	Fox and McDonald
<u>Case 40</u>	Water	2	0	No	0	Fox and McDonald
<u>Case 41</u>	Water	5	0	No	0	Fox and McDonald
<u>Case 42</u>	Water	2	0	No	0	Baumeister, Avallone and Baumeister
<u>Case 43</u>	Benzene	2	0	No	0	Baumeister, Avallone and Baumeister
<u>Case 44</u>	Ethyl Alcohol	3	0	No	0	Baumeister, Avallone and Baumeister
<u>Case 45</u>	Water	2	0	No	0	John and Haberman
<u>Case 46</u>	Water	7	0	No	0	John and Haberman
<u>Case 47</u>	Water	6	0	No	0	John and Haberman
<u>Case 48</u>	Water	2	0	No	0	John and Haberman
<u>Case 49</u>	Water	1	0	No	0	Janna

Summary of All Verification Models

Case	Fluid	Pipes	Pumps	NPSH	PRV 's	Reference
<u>Case 50</u>	Water	1	0	No	0	Janna
<u>Case 51</u>	Benzene	1	0	No	0	Janna
<u>Case 52</u>	Turpentine	9	0	No	0	Janna
<u>Case 53</u>	Water	4	0	No	0	Janna
<u>Case 54</u>	Water	2	0	No	0	Janna
<u>Case 55</u>	Water	2	1	No	0	Janna
<u>Case 56</u>	Fuel Oil	1	0	No	0	Chopey
<u>Case 57</u>	Kerosene	1	0	No	0	Chopey
<u>Case 58</u>	Water	6	3	No	0	Chopey
<u>Case 59</u>	Water	10	3	No	0	Jeppson
<u>Case 60</u>	Water	9	1	No	1	Jeppson
<u>Case 61</u>	Water	17	1	No	0	Jeppson
<u>Case 62</u>	Water	48	0	No	0	Jeppson
<u>Case 63</u>	Water	8	1	No	0	Jeppson
<u>Case 64</u>	Water	10	1	No	1	Jeppson
<u>Case 65</u>	Water	12	0	No	2	Jeppson
<u>Case 66</u>	Water	16	1	No	1	Jeppson
<u>Case 67</u>	Water	31	3	No	0	Jeppson

Summary of All Verification Models

Case	Fluid	Pipes	Pumps	NPSH	PRV 's	Reference
<u>Case 68</u>	Water	68	5	No	0	Jeppson
<u>Case 69</u>	Water	70	5	No	0	Jeppson
<u>Case 70</u>	Settling Slurry (Water/Sand)	1	0	No	0	Wilson, Addie, Sellgren & Clift
<u>Case 71</u>	Settling Slurry (Water/Coal)	1	0	No	0	Wilson, Addie, Sellgren & Clift
<u>Case 72</u>	Settling Slurry (Water/Sand)	2	1	No	0	Wilson, Addie, Sellgren & Clift
<u>Case 73</u>	Settling Slurry (Water/Ore)	2	1	No	0	Wilson, Addie, Sellgren & Clift
<u>Case 74</u>	Settling Slurry (Water/ Sand)	2	1	No	0	Wilson, Addie, Sellgren & Clift
<u>Case 75</u>	Settling Slurry (Water/ Sand)	1	0	No	0	Wilson, Addie, Sellgren & Clift
<u>Case 76</u>	Settling Slurry (Water/ Sand)	1	0	No	0	Wilson, Addie, Sellgren & Clift
<u>Case 77</u>	Methanol	1	0	No	0	Janna
<u>Case 78</u>	Water	2	1	No	0	John & Haberman
<u>Case 79</u>	Water	1	0	No	0	Fox & McDonald
<u>Case 80</u>	Air	4	0	No	0	Lindeburg

Summary of Verification Models with Pumps

[Summary of All Verification Models](#)

Comparison of AFT Fathom predictions for pumped systems to the published calculation results.

Case	Fluid	Pipes	Pumps	NPSH	PRV's	Reference
Case 3	Water	2	1	No	0	Miller
Case 11	Water	8	1	No	0	Brater, Williams, Lindell and Wei
Case 12	Water	3	1	Yes	0	Ingersoll-Dresser Pumps
Case 13	Water	3	1	Yes	0	Ingersoll-Dresser Pumps
Case 14	Water	3	1	Yes	0	Ingersoll-Dresser Pumps
Case 15	Water	3	1	Yes	0	Ingersoll-Dresser Pumps
Case 16	Water	3	1	Yes	0	Ingersoll-Dresser Pumps
Case 17	Water	2	1	No	0	Ingersoll-Dresser Pumps
Case 18	Specific gravity = 0.8	2	1	No	0	Karassik, Krutzsch, Fraser and Messina
Case 23	Viscous Fluid	2	1	No	0	Hydraulic Institute
Case 26	Water	8	1	Yes	0	Lindeburg
Case 55	Water	2	1	No	0	Janna
Case 58	Water	6	3	No	0	Chohey
Case 59	Water	10	3	No	0	Jeppson
Case 60	Water	9	1	No	1	Jeppson
Case 61	Water	17	1	No	0	Jeppson

Summary of Verification Models with Pumps

Case	Fluid	Pipes	Pumps	NPSH	PRV 's	Reference
<u>Case 63</u>	Water	8	1	No	0	Jeppson
<u>Case 64</u>	Water	10	1	No	1	Jeppson
<u>Case 66</u>	Water	16	1	No	1	Jeppson
<u>Case 67</u>	Water	31	3	No	0	Jeppson
<u>Case 68</u>	Water	68	5	No	0	Jeppson
<u>Case 69</u>	Water	70	5	No	0	Jeppson

Summary of Verification Models with NPSH

[Summary of All Verification Models](#)

Comparison of AFT Fathom predictions for pumped systems with NPSH requirements to the published calculation results.

Case	Fluid	Pipes	Pumps	NPSH	PRV's	Reference
Case 12	Water	3	1	Yes	0	Ingersoll-Dresser Pumps
Case 13	Water	3	1	Yes	0	Ingersoll-Dresser Pumps
Case 14	Water	3	1	Yes	0	Ingersoll-Dresser Pumps
Case 15	Water	3	1	Yes	0	Ingersoll-Dresser Pumps
Case 16	Water	3	1	Yes	0	Ingersoll-Dresser Pumps
Case 26	Water	8	1	Yes	0	Lindeburg

Summary of Verification Models with PRVs

[Summary of All Verification Models](#)

Comparison of AFT Fathom predictions for systems with PRVs to the published calculation results.

Case	Fluid	Pipes	Pumps	NPSH	PRV's	Reference
Case 60	Water	9	1	No	1	Jeppson
Case 64	Water	10	1	No	1	Jeppson
Case 65	Water	12	0	No	2	Jeppson
Case 66	Water	16	1	No	1	Jeppson

Summary of Verification Models with Slurries

[Summary of All Verification Models](#)

Comparison of AFT Fathom predictions for systems with slurries to the published calculation results.

Case	Fluid	Pipes	Pumps	NPSH	PRV's	Reference
Case 70	Settling Slurry (Water/Sand)	1	0	No	0	Wilson, Addie, Sellgren & Clift
Case 71	Settling Slurry (Water/Coal)	1	0	No	0	Wilson, Addie, Sellgren & Clift
Case 72	Settling Slurry (Water/Sand)	2	1	No	0	Wilson, Addie, Sellgren & Clift
Case 73	Settling Slurry (Water/Ore)	2	1	No	0	Wilson, Addie, Sellgren & Clift
Case 74	Settling Slurry (Water/Sand)	2	1	No	0	Wilson, Addie, Sellgren & Clift
Case 75	Settling Slurry (Water/Sand)	1	0	No	0	Wilson, Addie, Sellgren & Clift
Case 76	Settling Slurry (Water/Sand)	1	0	No	0	Wilson, Addie, Sellgren & Clift

Summary of Verification Models with ANS

[Summary of All Verification Models](#)

Comparison of AFT Fathom predictions for systems sized using the automated network sizing (ANS) module to published sizing results.

Case	Fluid	Pipes	Pumps	NPSH	PRV's	Reference
Case 77	Methanol	1	0	No	0	Janna
Case 78	Water	2	1	No	0	John & Haberman
Case 79	Water	1	0	No	0	Fox & McDonald
Case 80	Air	4	0	No	0	Lindeburg

Verification Cases

Verification Case 1

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify1.fth

REFERENCE: Perry's Chemical Engineers' Handbook, 7th Ed., 1997, McGraw-Hill, Robert H. Perry, Don W. Green, Eds., James N. Tilton, Ph.D., P.E., Author, Page 6-17, 18, Example 6

FLUID: Water

ASSUMPTIONS: N/A

RESULTS:

Parameter	Tilton	AFT Fathom
EGLinlet - Energy Gradeline at inlet (meters)	0.738	0.7397

DISCUSSION:

The flow conditions are specified in terms of velocity, at 2.0 m/s. Converting this to flow rate (for diameter of .0525 m) obtains 0.00433 cubic meters per second. The problem is stated in terms of flow requirement with an unknown inlet tank height. This is most easily modeled using an Assigned Flow junction at the inlet. The EGL (energy gradeline) at the inlet will yield the tank height.

The inlet loss factor of 0.5 is modeled as part of the inlet Assigned Flow.

The friction factor in the reference is the Fanning friction factor, whereas AFT Fathom uses the Moody friction factor. The two differ by a factor of 4. If this ratio is applied, results using either friction factor are the same.

The problem does not state the discharge boundary condition very clearly, but reading through the solution procedure it is evident that discharge condition is static pressure at atmospheric pressure. This is modeled using an Assigned Pressure junction at 1 atm.

The loss factor at the elbow, as developed in the reference, is 0.37.

The problem solution as given in the reference is 0.73 feet. However, if the numbers as given in the reference are multiplied, a value of 0.738 m results. The author apparently rounded the value down to 0.73.

[List of All Verification Models](#)

Verification Case 1 Problem Statement

[Verification Case 1](#)

Perry' Chemical Engineers' Handbook, 7th Ed., 1997, McGraw-Hill, Robert H. Perry, Don W. Green, Eds., James N. Tilton, Ph.D., P.E., Author, Page 6-17, 18, Example 6

[Perry's Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Equation (6-93) agrees with experimental data (Kemblowski and Kiljanski, *Chem. Eng. J. (Lausanne)*, **9**, 141-151 [1975]) for $\alpha < 11^\circ$. For Newtonian liquids, Eq. (6-93) simplifies to

$$\Delta p = \mu \left(\frac{8V_z}{D_2} \right) \left\{ \frac{1}{6 \tan(\alpha/2)} \left[1 - \left(\frac{D_2}{D_1} \right)^3 \right] \right\} \quad (6-94)$$

For creeping flow through rectangular or two-dimensional converging channels, the differential form of the Hagen-Poiseuille equation with equivalent diameter given by Eq. (6-49) may be used, provided the convergence is gradual.

Expansion and Exit Losses For ducts of any cross section, the frictional loss for a **sudden enlargement** (Fig. 6-13c) with turbulent flow is given by the Borda-Carnot equation:

$$l_e = \frac{V_1^2 - V_2^2}{2} = \frac{V_1^2}{2} \left(1 - \frac{A_1}{A_2} \right)^2 \quad (6-95)$$

where V_1 = velocity in the smaller duct
 V_2 = velocity in the larger duct
 A_1 = cross-sectional area of the smaller duct
 A_2 = cross-sectional area of the larger duct

Equation (6-95) is valid for incompressible flow. For compressible flows, see Benedict, Wyler, Dudek, and Glead (*J. Eng. Power*, **98**, 327-334 [1976]). For an infinite expansion, $A_1/A_2 = 0$, Eq. (6-95) shows that the exit loss from a pipe is 1 velocity head. This result is easily deduced from the mechanical energy balance Eq. (6-90), noting that $p_1 = p_2$. This exit loss is due to the dissipation of the discharged jet; there is no pressure drop at the exit.

For creeping Newtonian flow ($Re < 1$), the frictional loss due to a sudden enlargement should be obtained from the same equation for a sudden contraction (Eq. (6-92)). Note, however, that Boger, Gupta, and Tanner (ibid.) give an exit friction equivalent length of 0.12 diameter, increasing for power law fluids as the exponent decreases. For laminar flows at higher Reynolds numbers, the pressure drop is twice that given by Eq. (6-95). This results from the velocity profile factor α in the mechanical energy balance being 2.0 for the parabolic laminar velocity profile.

If the transition from a small to a large duct of any cross-sectional shape is accomplished by a **uniformly diverging duct** (see Fig. 6-13d) with a straight axis, the total frictional pressure drop can be computed by integrating the differential form of Eq. (6-89), $dl_e/dx = 2fV^2/D$ over the length of the expansion, provided the total angle α between the diverging walls is less than 7° . For angles between 7° and 45° , the loss coefficient may be estimated as 2.6 $\sin(\alpha/2)$ times the loss coefficient for a sudden expansion; see Hooper (*Chem. Eng.*, Nov. 7, 1988). Gibson (*Hydraulics and Its Applications*, 5th ed., Constable, London 1952, p. 93) recommends multiplying the sudden enlargement loss by 0.13 for $5^\circ < \alpha < 7.5^\circ$ and by $0.0110\alpha^{1.22}$ for $7.5^\circ < \alpha < 35^\circ$. For angles greater than 35° to 45° , the losses are normally considered equal to those for a sudden expansion, although in some cases the losses may be greater. Expanding flow through standard pipe reducers should be treated as sudden expansions.

Trumpet-shaped enlargements for turbulent flow designed for constant decrease in velocity head per unit length were found by Gibson (ibid., p. 95) to give 20 to 60 percent less frictional loss than straight taper pipes of the same length.

A special feature of expansion flows occurs when **viscoelastic liquids** are extruded through a die at a low Reynolds number. The extrudate may expand to a diameter several times greater than the die diameter, whereas for a Newtonian fluid the diameter expands only 10 percent. This phenomenon, called **die swell**, is most pronounced with short dies (Graessley, Glasscock, and Crawley, *Trans. Soc. Rheol.*, **14**, 519-544 [1970]). For velocity distribution measurements near the die exit, see Goulden and MacSparran (*J. Non-Newtonian Fluid Mech.*, **1**, 183-198 [1976]) and Whipple and Hill (*AIChE J.*, **24**, 654-671 [1978]). At high flow rates, the extrudate becomes distorted, suffering **melt fracture** at wall shear stresses greater than 10^5 N/m². This phenomenon is reviewed by Denn (*Ann. Review Fluid Mech.*, **22**, 13-34 [1990]). Ramamurthy (*J. Rheol.*, **30**, 337-357 [1986]) has found a dependence of apparent stick-slip behavior in melt fracture to be dependent on the material of construction of the die.

Fittings and Valves For **turbulent flow**, the frictional loss for fittings and valves can be expressed by the equivalent length or velocity head methods. As fitting size is varied, K values are relatively more constant than L_e/D values, but since fittings generally do not achieve geometric similarity between sizes, K values tend to decrease with increasing fitting size. Table 6-4 gives K values for many types of fittings and valves.

Manufacturers of valves, especially control valves, express valve capacity in terms of a flow coefficient C_v , which gives the flow rate through the valve in gal/min of water at 60°F under a pressure drop of 1 lb/in². It is related to K by

$$C_v = \frac{C_1 d^2}{\sqrt{K}} \quad (6-96)$$

where C_1 is a dimensional constant equal to 29.9 and d is the diameter of the valve connections in inches.

For **laminar flow**, data for the frictional loss of valves and fittings are meager (Beck and Miller, *J. Am. Soc. Nav. Eng.*, **56**, 62-83 [1944]; Beck, ibid., **56**, 235-271, 366-388, 389-395 [1944]; De Craene, *Heat. Piping Air Cond.*, **27**[10], 94-95 [1955]; Karr and Schutz, *J. Am. Soc. Nav. Eng.*, **52**, 239-256 [1940]; and Kittredge and Rowley, *Trans. ASME*, **79**, 1759-1766 [1957]). The data of Kittredge and Rowley indicate that K is constant for Reynolds numbers above 500 to 2,000, but increases rapidly as Re decreases below 500. Typical values for K for laminar flow Reynolds numbers are shown in Table 6-5.

Methods to calculate losses for **tee and wye junctions** for dividing and combining flow are given by Miller (*Internal Flow Systems*, 2d ed., Chap. 13, BHRA, Cranfield, 1990), including effects of Reynolds number, angle between legs, area ratio, and radius. Junctions with more than three legs are also discussed. The sources of data for the loss coefficient charts are Blaisdell and Manson (*U.S. Dept. Agric. Res. Serv. Tech. Bull.* 1283 [August 1963]) for combining flow and Gardel (*Bull. Tech. Suisse Romande*, **85**[9], 123-130 [1957]; **85**[10], 143-148 [1957]) together with additional unpublished data for dividing flow.

Miller (*Internal Flow Systems*, 2d ed., Chap. 13, BHRA, Cranfield, 1990) gives the most complete information on losses in **bends and curved pipes**. For turbulent flow in circular cross-section bends of constant area, as shown in Fig. 6-14a, a more accurate estimate of the loss coefficient K than that given in Table 6-4 is

$$K = K^* C_{bc} C_f \quad (6-97)$$

where K^* , given in Fig. 6-14b, is the loss coefficient for a smooth-walled bend at a Reynolds number of 10^6 . The Reynolds number correction factor C_{bc} is given in Fig. 6-14c. For $0.7 < r/D < 1$ or for $K^* < 0.4$, use the C_{bc} value for $r/D = 1$. Otherwise, if $r/D < 1$, obtain C_{bc} from

$$C_{bc} = \frac{K^*}{K^* + 0.2(1 - C_{bc, r/D=1})} \quad (6-98)$$

The correction C_f (Fig. 6-14d) accounts for the extra losses due to developing flow in the outlet tangent of the pipe, of length L_o . The total loss for the bend plus outlet pipe includes the bend loss K plus the straight pipe frictional loss in the outlet pipe $4fL_o/D$. Note that $C_f = 1$ for L_o/D greater than the termination of the curves on Fig. 6-14d, which indicate the distance at which fully developed flow in the outlet pipe is reached. Finally, the roughness correction is

$$C_f = \frac{f_{rough}}{f_{smooth}} \quad (6-99)$$

where f_{rough} is the friction factor for a pipe of diameter D with the roughness of the bend, at the bend inlet Reynolds number. Similarly, f_{smooth} is the friction factor for smooth pipe. For $Re > 10^6$ and $r/D \geq 1$, use the value of C_f for $Re = 10^6$.

Example 6: Losses with Fittings and Valves It is desired to calculate the liquid level in the vessel shown in Fig. 6-15 required to produce a discharge velocity of 2 m/s. The fluid is water at 20°C with $\rho = 1,000$ kg/m³ and $\mu = 0.001$ Pa·s, and the butterfly valve is at $\theta = 10^\circ$. The pipe is 2-in Schedule 40, with an inner diameter of 0.0525 m. The pipe roughness is 0.046 mm. Assuming the flow is turbulent and taking the velocity profile factor $\alpha = 1$, the engineering Bernoulli equation Eq. (6-16), written between surfaces 1 and 2, where the

6-18 FLUID AND PARTICLE DYNAMICS

TABLE 6-4 Additional Frictional Loss for Turbulent Flow through Fittings and Valves*

Type of fitting or valve	Additional friction loss, equivalent no. of velocity heads, K
45° ell, standard ^{b,c,d,e,f}	0.35
45° ell, long radius ^c	0.2
90° ell, standard ^{b,c,d,e,f,h}	0.75
Long radius ^{b,c,d,e}	0.45
Square or miter ^h	1.3
180° bend, close return ^{b,c,e}	1.5
Tee, standard, along run, branch blanked off ⁱ	0.4
Used as ell, entering run ^d	1.0
Used as ell, entering branch ^{c,d}	1.0
Branching flow ^{i,k}	1 ^l
Coupling ^{c,e}	0.04
Union ^c	0.04
Gate valve, ^{b,c,m} open	0.17
3/4 open ⁿ	0.9
1/2 open ⁿ	4.5
1/4 open ⁿ	24.0
Diaphragm valve, ^c open	2.3
3/4 open ⁿ	2.6
1/2 open ⁿ	4.3
1/4 open ⁿ	21.0
Globe valve, ^{c,m}	
Bevel seat, open	6.0
1/4 open ⁿ	9.5
Composition seat, open	6.0
1/2 open ⁿ	8.5
Plug disk, open	9.0
3/4 open ⁿ	13.0
1/2 open ⁿ	36.0
1/4 open ⁿ	112.0
Angle valve, ^{b,c} open	2.0
Y or blowoff valve, ^{b,m} open	3.0
Plug cock ^p	
θ = 5°	0.05
θ = 10°	0.20
θ = 20°	1.56
θ = 40°	17.3
θ = 60°	206.0
Butterfly valve ^q	
θ = 5°	0.24
θ = 10°	0.52
θ = 20°	1.54
θ = 40°	10.8
θ = 60°	118.0
Check valve, ^{b,c,m} swing	2.0 ^r
Disk	10.0 ^r
Ball	70.0 ^r
Foot valve ^r	15.0
Water meter, ^s disk	7.0 ^r
Piston	15.0 ^r
Rotary (star-shaped disk)	10.0 ^r
Turbine-wheel	6.0 ^r

*Lapple, *Chem. Eng.*, 56(5), 96-104 (1949), general survey reference.
^bFlow of Fluids through Valves, Fittings, and Pipes," Tech. Pap. 410, Crane Co., 1969.
^cFreeman, *Experiments upon the Flow of Water in Pipes and Pipe Fittings*, American Society of Mechanical Engineers, New York, 1941.
^dGiesecke, *J. Am. Soc. Heat Vent. Eng.*, 32, 461 (1926).
^e*Pipe Friction Manual*, 3d ed., Hydraulic Institute, New York, 1961.
^fItto, *J. Basic Eng.*, 82, 131-143 (1960).
^gGiesecke and Badgett, *Heat. Piping Air Cond.*, 4(6), 443-447 (1932).
^hSchoder and Dawson, *Hydraulics*, 2d ed., McGraw-Hill, New York, 1934, p. 213.
ⁱHoopes, Isakoff, Clarke, and Drew, *Chem. Eng. Prog.*, 44, 691-696 (1948).
^jGilman, *Heat. Piping Air Cond.*, 27(4), 141-147 (1955).
^kMcNown, *Proc. Am. Soc. Civ. Eng.*, 79, Separate 258, 1-22 (1953); discussion, *ibid.*, 80, Separate 396, 19-45 (1954). For the effect of branch spacing on junction losses in dividing flow, see Hecker, Nystrom, and Qureshi, *Proc. Am. Soc. Civ. Eng., J. Hydraul. Div.*, 103(HY3), 265-279 (1977).
^lThis is pressure drop (including friction loss) between run and branch, based on velocity in the mainstream before branching. Actual value depends on the flow split, ranging from 0.5 to 1.3 if mainstream enters run and from 0.7 to 1.5 if mainstream enters branch.
^mLansford, *Loss of Head in Flow of Fluids through Various Types of 1 1/2-in. Valves*, Univ. Eng. Exp. Sta. Bull. Ser. 340, 1943.

pressures are both atmospheric and the fluid velocities are 0 and $V = 2$ m/s, respectively, and there is no shaft work, simplifies to

$$gZ = \frac{V^2}{2} + l_e$$

Contributing to l_e are losses for the entrance to the pipe, the three sections of straight pipe, the butterfly valve, and the 90° bend. Note that no exit loss is used because the discharged jet is outside the control volume. Instead, the $V^2/2$ term accounts for the kinetic energy of the discharging stream. The Reynolds number in the pipe is

$$Re = \frac{DV\rho}{\mu} = \frac{0.0525 \times 2 \times 1000}{0.001} = 1.05 \times 10^5$$

From Fig. 6-9 or Eq. (6-38), at $e/D = 0.046 \times 10^{-3}/0.0525 = 0.00088$, the friction factor is about 0.0054. The straight pipe losses are then

$$l_{\text{pipe}} = \left(\frac{4fL}{D} \right) \frac{V^2}{2}$$

$$= \left(\frac{4 \times 0.0054 \times (1 + 1 + 1)}{0.0525} \right) \frac{V^2}{2}$$

$$= 1.23 \frac{V^2}{2}$$

The losses from Table 6-4 in terms of velocity heads K are $K = 0.5$ for the sudden contraction and $K = 0.52$ for the butterfly valve. For the 90° standard radius ($r/D = 1$), the table gives $K = 0.75$. The method of Eq. (6-94), using Fig. 6-14, gives

$$K = K^* C_{\text{re}} C_f$$

$$= 0.24 \times 1.24 \times 1.0 \times \left(\frac{0.0054}{0.0044} \right)$$

$$= 0.37$$

This value is more accurate than the value in Table 6-4. The value $f_{\text{smooth}} = 0.0044$ is obtainable either from Eq. (6-37) or Fig. 6-9.

The total losses are then

$$l_e = (1.23 + 0.5 + 0.52 + 0.37) \frac{V^2}{2} = 2.62 \frac{V^2}{2}$$

and the liquid level Z is

$$Z = \frac{1}{g} \left(\frac{V^2}{2} + 2.62 \frac{V^2}{2} \right) = 3.62 \frac{V^2}{2g}$$

$$= \frac{3.62 \times 2^2}{2 \times 9.81} = 0.73 \text{ m}$$

Curved Pipes and Coils For flow through curved pipe or coil, a secondary circulation perpendicular to the main flow called the **Dean effect** occurs. This circulation increases the friction relative to straight pipe flow and stabilizes laminar flow, delaying the transition Reynolds number to about

$$Re_{\text{crit}} = 2,100 \left(1 + 12 \sqrt{\frac{D_c}{D_c}} \right) \quad (6-100)$$

where D_c is the coil diameter. Equation (6-100) is valid for $10 < D_c/D < 250$. The **Dean number** is defined as

$$De = \frac{Re}{(D_c/D)^{3/2}} \quad (6-101)$$

In laminar flow, the friction factor for curved pipe f_c may be expressed in terms of the straight pipe friction factor $f = 16/Re$ as (Hart, *Chem. Eng. Sci.*, 43, 775-783 (1988))

TABLE 6-5 Additional Frictional Loss for Laminar Flow through Fittings and Valves

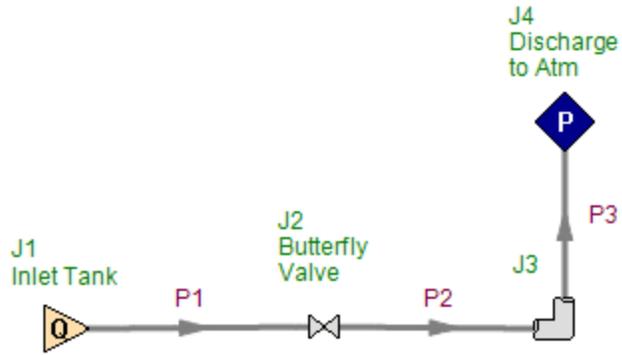
Type of fitting or valve	Additional frictional loss expressed as K			
	Re = 1,000	500	100	50
90° ell, short radius	0.9	1.0	7.5	16
Gate valve	1.2	1.7	9.9	24
Globe valve, composition disk	11	12	20	30
Plug	12	14	19	27
Angle valve	8	8.5	11	19
Check valve, swing	4	4.5	17	55

SOURCE: From curves by Kittredge and Rowley, *Trans. Am. Soc. Mech. Eng.*, 79, 1759-1766 (1957).

View Verification Case 1 Model

[Verification Case 1](#)

Verification Case 1



Verification Case 2

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: [FthVerify2.fth](#)FthVerify2.fth

REFERENCE: Piping Handbook, 7th Ed., 2000, McGraw-Hill, Mohinder L. Nayyar, P.E., Ed., Tadeusz J. Swierzawski, Author, Page B.375-381, Example B8.1

FLUID: Water

ASSUMPTIONS: N/A

RESULTS:

Parameter	Swierzawski	AFT Fathom
Pressure drop (psid)	61.48	61.33

DISCUSSION:

The exit loss factor of 1.0 is modeled as part of the exit Assigned Flow.

The overall pressure drop in AFT Fathom is obtained by subtracting the discharge pressure (438.67 psia) from the supply (500 psia). It is also displayed in the General Results list at the top of the Output window. Finally, the pressure difference is given in the Junction Deltas table at the top of the Output window. The junction delta was setup in the Output Control window.

On page B.380 the Piping Handbook calculates the Flow Resistance Coefficients as 9.77 as shown in the table on that page. However, if one adds up the individual K factors, 9.84 is obtained, not 9.77. To better match the overall resistance assumed for the handbook calculation, the tee K factor was therefore reduced from 0.33 to 0.26 so that a K factor of 9.77 is obtained.

The difference in the final answer (61.48 vs. 61.33) is mostly due to round-off errors in the handbook calculations. If more digits are saved in the handbook calculations, the answer will be much closer to 61.33.

[List of All Verification Models](#)

Verification Case 2 Problem Statement

[Verification Case 2](#)

Piping Handbook, 7th Ed., 2000, McGraw-Hill, Mohinder L. Nayyar, P.E., Ed., Tadeusz J. Swierzawski, Author, Page B.375-381, Example B8.1

[Nayyar Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

FLOW OF FLUIDS		B.375
$d2/d1$	$(d1/d2)^2$	
0.99	1.050	
0.98	1.106	
0.97	1.165	
0.96	1.230	
0.95	1.292	
0.90	1.694	
0.80	3.052	
0.70	5.950	

in Fig. B8.7. In addition, there is the cost of the pump and the pump drive, which increases as greater pumping power is required, as represented by cost c .

The total cost, which is the sum of these three costs ($T_{\text{COST}} = a + b + c$) in \$/year, is shown on Fig. B8.7 as reaching a minimum value at the optimum flow velocity. The analysis for each piping system should consider the optimization of flow velocity (optimum internal diameter) of the pipe under consideration. It is important to note that cost b depends strongly on the plant operating mode or the load factor, and on other economic indicators for a particular project. Industry data containing updated prices of equipment, piping, and labor are needed to implement this optimization procedure.

Sample Problem B8.1. What is the pressure drop $p_A - p_B$ (see Fig. B8.8), when water at 200°F (93.3°C) flows in a piping system at the rate of $\dot{m} = 450,200$ lb_m/h (204,207 kg/h), $p_A = 500$ psia (34.47 bar)?

Pipe Data. For 6-in nominal size, schedule 40 pipe, the internal diameter $d = 6.065$ in, $D = 0.5054$ ft, $A = 0.200$ ft².

Properties of Fluid. From the ASME Steam Tables software (Ref. 5) for $T = 200^\circ\text{F}$ and $p = 500$ psia, the dynamic viscosity of water (see attached computer printout, Table B8.9) is

$$\mu = 63.43 * 10^{-7} \text{ (lb}_f \cdot \text{s) / ft}^2$$

From the same computer printout, the specific volume of water at $T = 200^\circ\text{F}$ and $p = 500$ psia is

$$v = 0.01661 \text{ ft}^3/\text{lb}_m$$

Then, the kinematic viscosity of water is [see Eq. (B8.3)]:

$$\begin{aligned} \nu &= \frac{\mu}{\rho} g_c = (63.43 * 10^{-7}) (0.01661) (32.174) \\ &= 3.39 * 10^{-6} \text{ ft}^2/\text{s} \end{aligned}$$

The flow velocity is calculated from the continuity equation

$$\begin{aligned} w &= \frac{\dot{m}}{\rho A} \\ \text{where } \dot{m} &= \frac{450,200}{3600} = 125.06 \text{ lb}_m/\text{s} \end{aligned}$$

Verification Case 2 Problem Statement

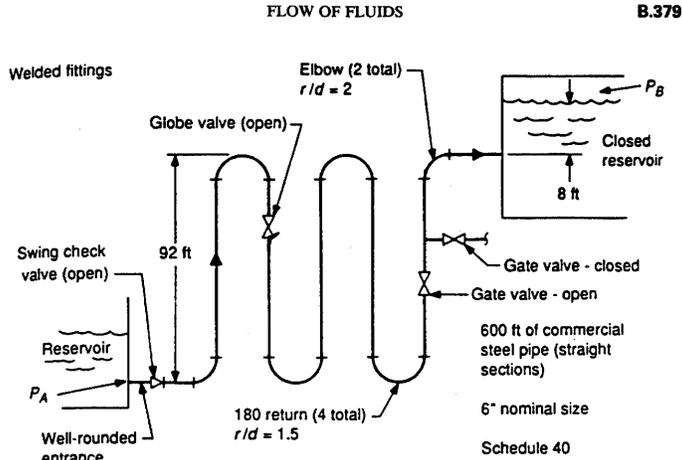


FIGURE B8.8 Reference for sample problem B8.1.

TABLE B8.9 Computer Printout of Steam/Water Properties

STEAM PROPERTIES	INPUT	*	RESULTS
-----	-----		-----
ABS. PRESSURE, bar	.0000		34.4738
psia	500.0000		500.0000
-----	-----	*	-----
TEMPERATURE, C	.0000		93.3333
F	200.0000		200.0000
-----	-----	*	-----
ENTHALPY, kJ/kg	.0000		393.5460
Btu/lbm	.0000		169.1943
-----	-----	*	-----
ENTROPY, kJ/(kg.K)	.000000		1.228453
Btu/(lbm.R)	.000000		.293411
-----	-----	*	-----
SPEC. VOLUME m ³ /kg	.000000		.001037
ft ³ /lbm	.000000		.016609
-----	-----	*	-----
QUALITY, decimal	.000000		-1.000000
-----	-----	*	-----
DYNAM. VISC N.s/m ²	.000000000		.000303681
lbf.s/ft ²	.000000000		.000006343
-----	-----	*	-----
UNITS FOR INPUT: UNT = 1-SI, 2-BRITISH			* 1995/TJS
UNT	PRESSURE	TEMPERAT	ENTHALPY
		ENTROPY	QUALITY

Verification Case 2 Problem Statement

B.380 GENERIC DESIGN CONSIDERATIONS

Then:

$$w = \frac{125.06}{0.200} 0.01661 = 10.39 \text{ ft/s}$$

Using Eq. (B8.16), the Reynolds number is

$$\text{Re} = \frac{Dw}{\nu} = \frac{(0.5054)(10.39)}{3.39 \times 10^{-6}} = 1.549 \times 10^6$$

D'Arcy-Weisbach Friction Factor. For commercial steel pipe from Ref. 7, the relative roughness is

$$\frac{\varepsilon}{D} = 0.00015/0.5054 = 0.0003$$

and the friction factor in zone of complete turbulence is

$$f_r = 0.015$$

From Eq. (B8.17), using, for example, the Newton-Raphson method for solving equations [or using the Moody diagram, Fig. (B8.5)], for calculated values of Re and ε/D , the *D'Arcy-Weisbach* friction factor, f , is found:

$$f = 0.0154$$

Flow Resistance Coefficients⁷:

Local disturbance	$K(f_r = 0.015)$
Entrance	$K = 0.04$
4 180° turns	$(4)(22.18 f_r) = 1.33$
1 globe valve (open)	$340 f_r = 5.10$
1 tee flow through run	$20 f_r = 0.33$
1 swing check valve (open)	$100 f_r = 1.50$
2 long radius elbows	$(2)(14f_r) = 0.42$
1 gate valve (open)	$8 f_r = 0.12$
Exit	$K = 1.00$
	$\sum K = 9.77$

Total Resistance

$$\begin{aligned} K_{\text{tot}} &= \sum K + f \frac{L}{D} \\ &= 9.77 + 0.0154 \frac{600}{0.5054} \\ &= 28.05 \end{aligned}$$

“Friction” Pressure Drop in Pipe. From Eqs. (B8.14) and (B8.18):

$$\begin{aligned} \Delta p &= K_{\text{tot}} \rho \frac{w^2}{2g_c} \\ &= 28.05 \frac{(10.39)^2}{(0.01661)(2)(32.174)} \\ &= 2833.08 \frac{\text{lb}_f}{\text{ft}^2} \\ &= \frac{2833.08}{144} = 19.67 \text{ psi} \end{aligned}$$

Pressure Difference $p_A - p_B$. From Bernoulli's equation (Eq. B8.12 with $H_f = 0$ and $H_p = 0$):

$$\begin{aligned} \frac{\rho_A w_A^2}{2g_c} + p_A + \frac{g}{g_c} z_A \rho_A \\ = \frac{\rho_B w_B^2}{2g_c} + p_B + \frac{g}{g_c} z_B \rho_B + \Delta p \end{aligned}$$

Assuming $\rho_A = \rho_B = \rho$ and $g = 32.174 \frac{\text{ft}}{\text{s}^2}$

$$\begin{aligned} p_A - p_B &= \frac{g}{g_c} (z_B - z_A) \frac{\rho}{144} + \Delta p \\ &= \frac{100}{(144)(0.01661)} + 19.67 = 61.48 \text{ psi (4.24 bar)} \end{aligned}$$

Applications: Oil and Other Liquid Systems

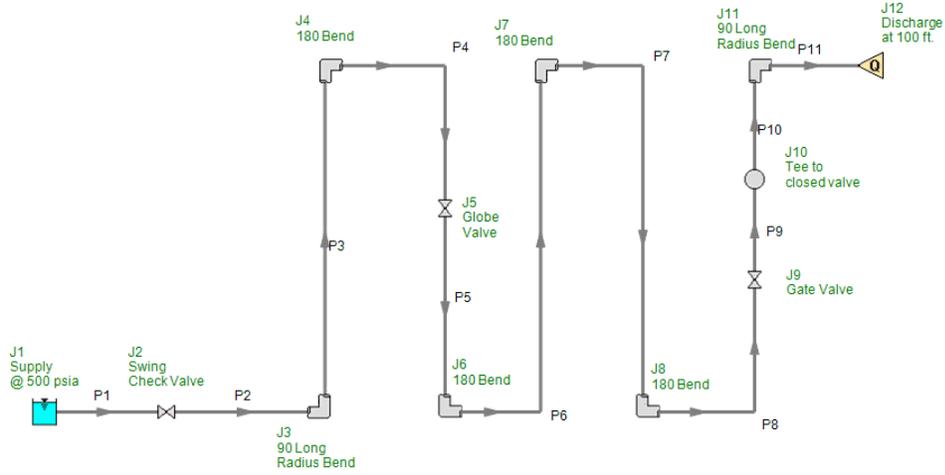
The calculation of oil flow through pipes is a much more complicated process than a similar calculation for water flow. Water properties are well-defined and many, including viscosity, are nearly constant within liquid temperature ranges, whereas oil is quite different. No two oils have the same physical properties, and any given oil is subject to important physical changes at expected variable temperatures. When considering oil flow in pipes, the most important variable physical property is viscosity.

Ordinary crude oil is not a homogeneous liquid; it is a very complex fluid composed of compounds of carbon and hydrogen which exist in petroleum in a wide variety of combinations. Physical properties change accordingly, and they also change as a result of temperature variations. Therefore, this handbook assumes that the user will research the fluid of interest in other sourcebooks (such as the *Petroleum Processing Handbook*⁹) to determine important properties including density and viscosity.

The “Viscosity” subsection in this handbook refers to Tables B8.5 and B8.6 for conversion among several common units of viscosity. The special names and symbols for derived CGS units (such as dyne, erg, poise, and stokes) should not be used

View Verification Case 2 Model

[Verification Case 2](#)



Verification Case 3

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify3.fth

REFERENCE: D.S. Miller, Internal Flow Systems, 2nd Ed., 1990, Gulf Publishing Co., Page 28-29

FLUID: Water

ASSUMPTIONS: Size pump for given flow

RESULTS:

Parameter	Miller	AFT Fathom
System head loss (meters)	21.6	21.51
Required pump head rise (meters)	28.6	28.51
Required pump pressure rise (bar)	2.8	2.796

DISCUSSION:

The reflux valve K value of 0.5 was lumped into pipe 2 as a fitting and loss value.

The overall system head loss in AFT Fathom is obtained by subtracting the discharge EGL (1.5 m) from the supply EGL (8.5 m), then subtracting this difference from the pump head rise (28.51 m).

The pump head rise and pressure rise in AFT Fathom is given in the pump summary at the top of the Output window, and also in the junction output table in the lower part of the window.

There is some slight disagreement with Miller due to round-off errors.

[List of All Verification Models](#)

Verification Case 3 Problem Statement

Verification Case 3

D.S. Miller, Internal Flow Systems, 2nd Ed., 1990, Gulf Publishing Co., Page 28-29

Miller Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

28

Internal flow systems

and D is the hydraulic diameter or pipe diameter. For a passage D is given by

$$D = \frac{4 \times \text{Passage cross-sectional area}}{\text{Passage perimeter}}$$

The loss coefficients, K , in equation (3.1) are provided as a function of the Reynolds number. The coefficients can be found from Part 2 or from additional sources.

The overall system head loss, H , or pressure loss, P , is found by summing the head and pressure losses for individual components. For a system with n components

$$H_{\text{overall}} = \sum_{i=1}^{i=n} \Delta H_i \quad (3.2a)$$

$$P_{\text{overall}} = \sum_{i=1}^{i=n} \Delta P_i = \rho g H_{\text{overall}} \quad (3.2b)$$

To obtain the duty of a pump or fan it is necessary to add the static lift or pressure difference across the ends of the system to the values obtained from equations (3.2a) or (3.2b).

3.3. SYSTEM CALCULATION AND PUMP OR FAN SELECTION

The method adopted for carrying out a calculation of overall pressure loss depends on individual preference and the suitability of the method for particular types of system. For instance, if the mean velocity is constant throughout the system, the individual loss coefficients could be summed and multiplied by the velocity head to obtain the overall head loss.

Complexities arise mainly from interaction between closely spaced components involving a departure from simple summing of individual component losses. For systems consisting mainly of straight pipes or passages, interaction effects are seldom important as regards pressure losses. If the distance between components is more than four diameters, neglecting interaction effects will usually result in the loss being slightly overestimated.

As an example of one calculation procedure, consider the simplified system in Fig. 3.1. For calculation purposes the components are best represented connected together at nodes as in Fig. 3.2, pipes being treated in the same manner as components. This type of system representation has advantages when carrying out computer solutions of complex systems and pipe networks.

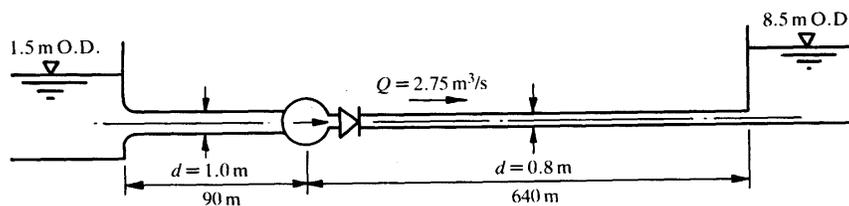


Fig. 3.1. Simple system

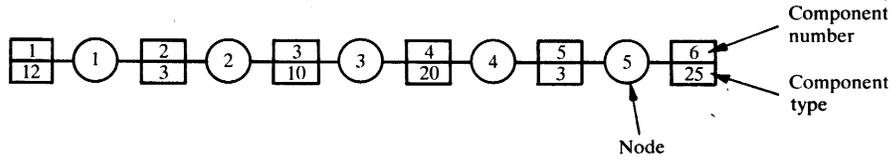


Fig. 3.2. Node and component representation of the system in Fig. 3.1

Table 3.1. Calculation for the system shown in Fig. 3.1

($\rho = 1000 \text{ kg/m}^3$, $\nu = 1.1 \times 10^{-6} \text{ m}^2/\text{s}$, friction coefficient taken as 0.015, including an allowance for fouling)

Node	Component		Diameter (m)	Velocity (m/s)	Reynolds number	$U^2/2g$ (m)	K^*	Head loss (m)	Pressure loss (kN/m^2)
	No.	Type							
1	1	12-reservoir inlet	1.0	3.50	3.2×10^6	0.625	0.1	0.06	0.589
1-2	2	3-pipe	1.0	3.50	3.2×10^6	0.625	1.35	0.84	8.240
2-3	3	10-pump	—	—	—	—	—	—	—
3-4	4	20-reflux valve	0.8	5.47	4.0×10^6	1.53	0.5	0.76	7.456
4-5	5	3-pipe	0.8	5.47	4.0×10^6	1.53	12.0	18.36	180.112
5	6	25-reservoir outlet	0.8	5.47	4.0×10^6	1.53	1.0	1.53	15.000
Total loss =								21.55	211

*For pipes $K = fL/D$.

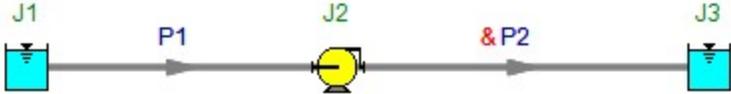
From Part 2 the loss coefficients, K , in Table 3.1 are appropriate for the various components of Fig. 3.1. Other relevant data are given in Table 3.1 to which, in some cases, it will be necessary to add details of pipe roughness and of interaction correction factors if components are close together. An example using interaction correction factors is included in Chapter 10.

$$\begin{aligned}
 \text{Pump head} &= \text{Total head loss} + \text{Static lift} \\
 &= 21.6 \text{ m} + 7 \text{ m} \\
 &= 28.6 \text{ m} \\
 &= 28.6 \times 1000 \times 9.81 = 280 \text{ kN/m}^2 = 2.8 \text{ bar}
 \end{aligned}$$

A pump or fan that most closely meets a design duty will usually be selected from a standard range. The head-flow characteristic of a suitable pump for the system in the example is shown in Fig. 3.3. Also shown in Fig. 3.3 is the system characteristic. For most purposes

View Verification Case 3 Model

[Verification Case 3](#)



Verification Case 4

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify4.fth

REFERENCE: D.S. Miller, Internal Flow Systems, 2nd Ed., 1990, Gulf Publishing Co., Page 197-199

FLUID: Air

ASSUMPTIONS: Air is incompressible

RESULTS:

Parameter	Miller	AFT Fathom
System head loss (m of air)	40.3	40.36
System head loss (mm of water)	49.6	49.64

DISCUSSION:

The velocity and diameter are specified. For use in AFT Fathom, this is converted to a flow rate, obtaining 0.8 m³/sec.

The head loss in the pipe in AFT Fathom can be easily obtained from the pipe dP Stag and dH output values.

[List of All Verification Models](#)

Verification Case 4 Problem Statement

Verification Case 4

D.S. Miller, Internal Flow Systems, 2nd Ed., 1990, Gulf Publishing Co., Page 197-199

Miller Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Friction in pipes and passages

197

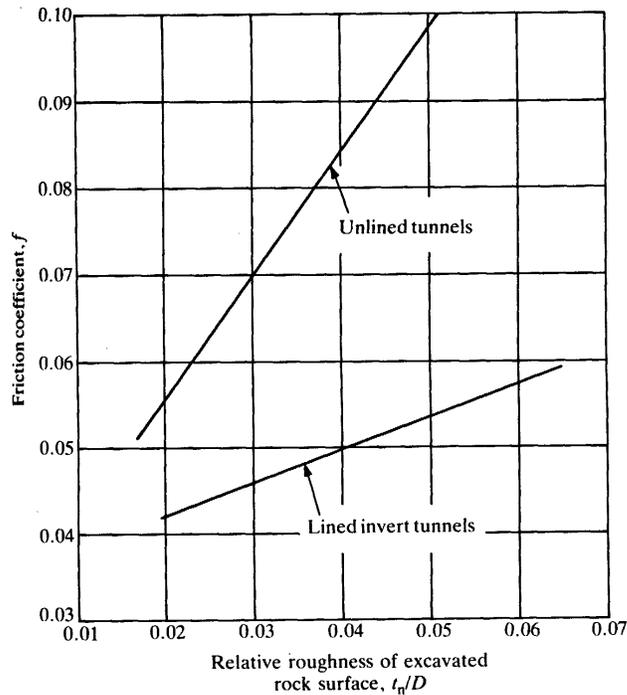


Fig. 8.5(1). Friction coefficients for rock tunnels

8.6. EXAMPLES

EXAMPLE 1: GIVEN Q , L , D , ν AND k , FIND THE HEAD LOSS

Calculate the head loss for air flowing in a 0.2 m square cross-section passage 25 m long, given:

$$Q = 0.8 \text{ m}^3/\text{s}$$

$$\text{Air density} = 1.23 \text{ kg/m}^3$$

$$\text{Kinematic viscosity} = 1.45 \times 10^{-5} \text{ m}^2/\text{s}$$

$$\text{Surface roughness, } k = 0.025 \text{ mm}$$

From equation (8.2)

$$D = 4A/P_r = 4 \times 0.2^2 / 4 \times 0.2 = 0.2 \text{ m}$$

Relative roughness

$$k/D = 0.025 \times 10^{-3} / 0.2 = 1.2 \times 10^{-4}$$

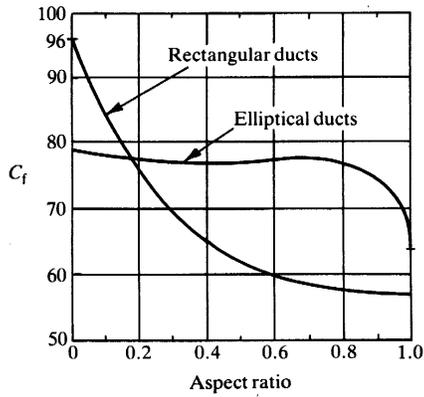


Fig. 8.6. Laminar flow coefficients — rectangular and elliptical cross-sections

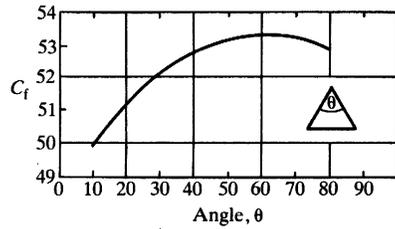


Fig. 8.7. Laminar flow coefficients — isosceles triangle

Mean velocity

$$U = Q/A = 0.8/0.2 \times 0.2 = 20 \text{ m/s}$$

Reynolds number

$$Re = DU/v = 0.2 \times 20/1.45 \times 10^{-5} = 2.8 \times 10^5$$

From equation (8.1) 4

$$f = 0.25 \left/ \left[\log \left(\frac{k}{3.7D} + \frac{5.74}{Re^{0.9}} \right) \right]^2 \right. = 0.25 \left/ \left[\log \left(\frac{1.2 \times 10^{-4}}{3.7} + \frac{5.74}{(2.8 \times 10^5)^{0.9}} \right) \right]^2 \right. = 0.0158$$

For a Reynolds number of 2.8×10^5 and a relative roughness of 1.2×10^{-4} the friction coefficient from Fig. 8.1 is also 0.0158.

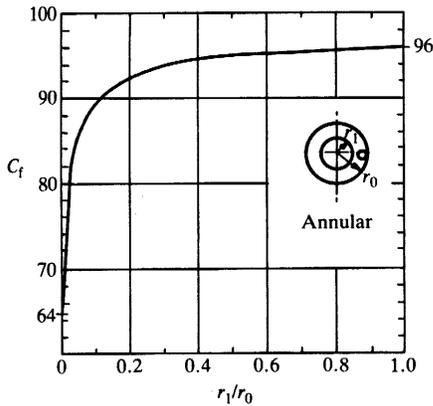


Fig. 8.8. Laminar flow coefficients — annular cross-sections

From equation (8.3)

$$K_f = fL/D = 0.0158 \times 25/0.2 = 1.975$$

Head loss

$$\begin{aligned} \Delta H &= K_f U^2 / 2g = 1.975 \times 20^2 / (2 \times 9.81) \\ &= 40.3 \text{ m of air} = 49.6 \text{ mm of water} \end{aligned}$$

EXAMPLE 2: GIVEN ΔH , L , D , ν AND k , FIND THE FLOW RATE Q

Water at 15°C flows by gravity between two tanks in a system consisting of 50 mm diameter pipes and components. The combined loss coefficient for the components is $K_f = 6.5$ for Reynolds numbers greater than 10^5 . If the total length of the straight pipes is 40 m, determine the flow for a head differential between the tanks of 6 m. Assume that the pipe is smooth. Kinematic viscosity = $1.14 \times 10^{-6} \text{ m}^2/\text{s}$.

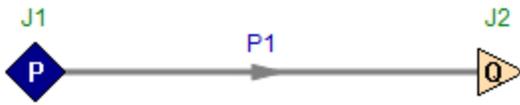
First trial

Assume $f = 0.015$, which lies in the middle of the smooth curve in Fig. 8.1.
Pipe loss coefficient

$$K_f = fL/D = 0.015 \times 40/50 \times 10^{-3} = 12$$

View Verification Case 4 Model

[Verification Case 4](#)



Verification Case 5

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify5.fth

REFERENCE: D.S. Miller, Internal Flow Systems, 2nd Ed., 1990, Gulf Publishing Co., Page 199-200

FLUID: Water

ASSUMPTIONS: N/A

RESULTS:

Parameter	Miller	AFT Fathom
Flow rate (m ³ /sec) x 1000	4.68	4.674

DISCUSSION:

The K factor of 6.5 is entered in pipe 1 as a fitting and loss value.

The answer differs slightly because Miller reads the friction factor (0.0179) off of a chart, whereas AFT Fathom uses a roughness of zero to obtain the friction factor (0.01784).

[List of All Verification Models](#)

Verification Case 5 Problem Statement

Verification Case 5

D.S. Miller, Internal Flow Systems, 2nd Ed., 1990, Gulf Publishing Co., Page 199-200

Miller Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Friction in pipes and passages

199

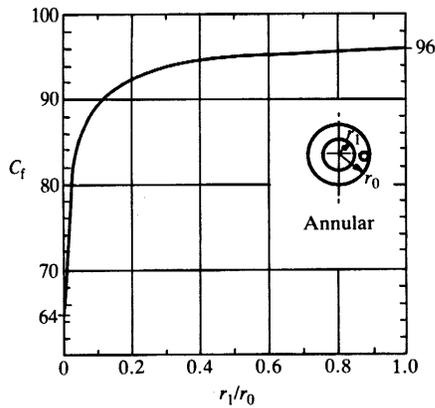


Fig. 8.8. Laminar flow coefficients — annular cross-sections

From equation (8.3)

$$K_t = fL/D = 0.0158 \times 25/0.2 = 1.975$$

Head loss

$$\begin{aligned} \Delta H &= K_t U^2 / 2g = 1.975 \times 20^2 / (2 \times 9.81) \\ &= 40.3 \text{ m of air} = 49.6 \text{ mm of water} \end{aligned}$$

EXAMPLE 2: GIVEN ΔH , L , D , ν AND k , FIND THE FLOW RATE Q

Water at 15°C flows by gravity between two tanks in a system consisting of 50 mm diameter pipes and components. The combined loss coefficient for the components is $K_t = 6.5$ for Reynolds numbers greater than 10^5 . If the total length of the straight pipes is 40 m, determine the flow for a head differential between the tanks of 6 m. Assume that the pipe is smooth. Kinematic viscosity = $1.14 \times 10^{-6} \text{ m}^2/\text{s}$.

First trial

Assume $f = 0.015$, which lies in the middle of the smooth curve in Fig. 8.1.

Pipe loss coefficient

$$K_t = fL/D = 0.015 \times 40/50 \times 10^{-3} = 12$$

Head loss

$$\Delta H = 6 \text{ m} = (K_f + K_c)U^2/2g$$

Mean velocity

$$U = \sqrt{\frac{6 \times 2 \times 9.81}{(12 + 6.5)}} = 2.52 \text{ m/s}$$

Reynolds number

$$\text{Re} = \frac{UD}{\nu} = \frac{2.52 \times 50 \times 10^{-3}}{1.14 \times 10^{-6}} = 0.111 \times 10^6$$

From Fig. 8.1

$$f = 0.0178$$

Second trial

Assume $f = 0.0178$

$$K_f = 12 \times 0.0178/0.015 = 14.2$$

$$U = 2.52 \sqrt{\frac{12 + 6.5}{14.2 + 6.5}} = 2.38 \text{ m/s}$$

$$\text{Re} = 0.111 \times 10^6 \times 2.38/2.52 = 0.105 \times 10^6$$

from Fig. 8.1, $f = 0.0179$, which agrees with the assumed value to within the accuracy of Fig. 8.1.

Flow

$$\begin{aligned} Q &= UA = 2.38 \times 0.785 \times 50^2 \times 10^{-6} \\ &= 4.68 \times 10^{-3} \text{ m}^3/\text{s} \end{aligned}$$

EXAMPLE 3: GIVEN ΔH , Q , L , ν AND k , FIND THE DIAMETER

0.58 m³ of water at 15°C has to flow through a constant diameter system with a head loss of not more than 3 m of water-gauge. The system consists of a number of sections of straight new steel pipe with a total length of 145 m, and a number of components. The estimated total loss coefficient for the components is $K_c = 2.9$ at Reynolds numbers greater than 10^6 . Determine the minimum pipe and component diameter required. Kinetic viscosity = 1.14×10^{-6} m²/s. From Table 8.1, hydraulic roughness, $k = 0.025$ mm.

First trial

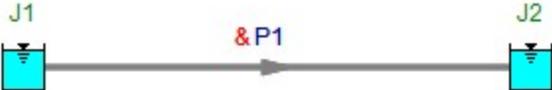
Assume diameter $D = 0.6$ m.

Mean velocity

$$U = Q/A = 0.58/(0.785 \times 0.6^2) = 2.052 \text{ m/s}$$

View Verification Case 5 Model

[Verification Case 5](#)



Verification Case 6

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify6.fth

REFERENCE: Ernest Brater, Horace Williams, James Lindell, C.Y. Wei, Handbook of Hydraulics, 7th Ed., 1996, McGraw-Hill, Page 6.30, Example 6.1a

FLUID: Water

ASSUMPTIONS: N/A

RESULTS:

Parameter	Handbook	AFT Fathom
Head loss (meters)	1.53	1.614

DISCUSSION:

The velocity and diameter are specified. For use in AFT Fathom, this is converted to a flow rate, obtaining 0.5655 m³/s.

The answers differ slightly because the handbook authors read the friction factor (0.015) off of a chart, whereas AFT Fathom solves for the friction factor using the Colebrook-White equation (obtaining 0.01583, 5.5% higher).

If the pipe is modeled as an explicit friction factor at 0.015, the exact same answer as the handbook is obtained.

No pressure boundary conditions are specified. In determining pressure drop, the boundary pressure plays no role. Therefore any value is suitable. The model assumes 1 MPa at the inlet.

[List of All Verification Models](#)

Verification Case 6 Problem Statement

[Verification Case 6](#)

Ernest Brater, Horace Williams, James Lindell, C.Y. Wei, Handbook of Hydraulics, 7th Ed., 1996, McGraw-Hill, Page 6.30, Example 6.1a

[Brater, Williams, Lindell and Wei Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

6.30

HANDBOOK OF HYDRAULICS

ferred to^{10,16} for corrugated-metal pipes resulted in values of f varying from 0.051 to 0.078, a range of 53 percent based on the smaller value, whereas for the same tests the values of n varied from 0.0229 to 0.0248, a range of only 8 percent based on the smaller value. Obviously, it is simpler and perhaps more accurate to use an n of 0.024 for all corrugated-metal pipes in the range of diameters and Reynolds numbers tested than to use the somewhat more cumbersome Darcy-Weisbach equation. Furthermore, extrapolation to other pipe sizes might also be more safely accomplished in this case by using the Manning equation. However, particularly for smoother pipe and large values of the Reynolds number, more dependable results can be obtained by means of the Darcy-Weisbach equation.

Although the developments described in the preceding section have vastly improved the understanding of the mechanics of energy dissipation in conduits, the problem is not yet fully understood, and the engineer should realize that the solution of all problems outside the laminar range must depend on the interpolation or extrapolation of experimental data determined under conditions which may have been dissimilar to that in which they will be used. For materials and pipe sizes commonly used, the curves of f and tables of n will give good results. When unusual surfaces or very large pipe sizes are involved, and especially for very long conduits in which the energy loss becomes a vital design consideration, the literature should be searched for tests on similar conduits. The U.S. Bureau of Reclamation has presented much information on friction factors.⁸ Other test results of interest have been reported by Burke⁹ and Elder.¹¹

The computation of energy loss will be illustrated by the following examples. Section 13, Examples 13.6 and 13.7, illustrate similar computations using a digital computer.

Example 6.1. Determine the energy loss in 300 m of new 60-cm concrete pipe when water at 21°C is flowing at an average velocity of 2 m/s (1) using the Darcy-Weisbach equation and (2) using the Manning equation.

1. Obtaining the kinematic viscosity from Table 1.2,

$$R = \frac{dV}{\nu} = \frac{0.60 \times 2.0}{0.984 \times 10^{-6}} = 1.2 \times 10^6$$

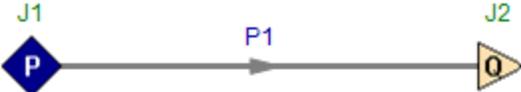
From Table 6.1, $\epsilon = 0.2$ mm. Then $\epsilon/d = 0.0002/0.60 = 0.00033$.

From Fig. 6.4, $f = 0.015$. Then, from Eq. (6.19),

$$h = f \frac{l}{d} \frac{V^2}{2g} = 0.015 \times \frac{300}{0.60} \times \frac{4}{19.6} = 1.53 \text{ m}$$

View Verification Case 6 Model

[Verification Case 6](#)



Verification Case 7

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify7.fth

REFERENCE: Ernest Brater, Horace Williams, James Lindell, C.Y. Wei, Handbook of Hydraulics, 7th Ed., 1996, McGraw-Hill, Page 6.31, Example 6.2

FLUID: Unspecified

ASSUMPTIONS: N/A

RESULTS:

Parameter	Handbook	AFT Fathom
Head loss (meters)	62.8	62.65

DISCUSSION:

The velocity and diameter are specified. For use in AFT Fathom, this is converted to a flow rate, obtaining 0.8482 cm³/sec. The unit weight is specified as 9580 N/m³. At 9.81 meters/sec² gravitational acceleration, the density is thus 976.55 kg/m³.

The answers differ slightly because the handbook rounds off the Reynolds number to 3 digits.

No pressure boundary conditions are specified. In determining pressure drop, the boundary pressure plays no role. Therefore any value is suitable. The model assumes 1 MPa at the inlet.

[List of All Verification Models](#)

Verification Case 7 Problem Statement

[Verification Case 7](#)

Ernest Brater, Horace Williams, James Lindell, C.Y. Wei, Handbook of Hydraulics, 7th Ed., 1996, McGraw-Hill, Page 6.31, Example 6.2

[Brater, Williams, Lindell and Wei Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

PIPES

6.31

2. From Table 6.4, minimum $n = 0.010$. Then from Eq. (6.26c),

$$h = \frac{6.25n^2 V^2}{d^{4/3}} = \frac{6.25 \times 0.010^2 \times 300 \times 4}{(0.60)^{4/3}} = 1.48 \text{ m}$$

Table 6.3 may also be used to obtain this solution. The discharge is obtained as follows:

$$Q = aV = 0.283 \times 2 = 0.565 \text{ m}^3/\text{s}$$

Then four-way interpolation in Table 6.3 for $n = 0.010$ gives $s = 0.0049$ and $sl = 1.47 \text{ m}$.

Example 6.2. Determine the energy loss in 15 m of 6-mm tubing for a liquid having a unit weight of 9580 N/m^3 and a viscosity of $1.50 \text{ N}\cdot\text{s/m}^2$ when the velocity is 3 cm/s .

$$R = \frac{dv \rho}{\mu} = \frac{0.006 \times 0.03 \times 9580}{1.5 \times 9.8} = 0.117$$

and from Eqs. (6.19) and (6.20),

$$h = f \frac{l}{d} \frac{V^2}{2g} = \frac{64}{R} \frac{l}{d} \frac{V^2}{2g} = \frac{64}{0.117} \times \frac{15}{0.006} \times \frac{0.03^2}{19.6} = 62.8 \text{ m}$$

MINOR LOSSES

Energy losses resulting from rapid changes in the direction or magnitude of the velocity are called *minor* losses, or *local* losses. The term minor loss is appropriate for pipelines that include long reaches of uniform straight pipe. However, for short pipes it is a misnomer, because the minor losses may be greater than the friction losses.

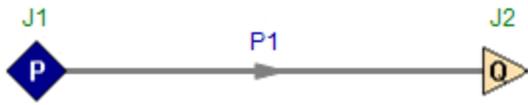
Minor losses are usually expressed in terms of the kinetic energy and a coefficient, as illustrated by the equation

$$h_m = K_m \frac{V^2}{2g} \quad (6.29)$$

When a change in pipe size occurs, two velocities are involved, and both may be included in the expression for energy loss, as will be shown for the case of enlargements and contractions. Under certain conditions, particularly for the compound-pipe problem (see Compound Pipes), it may be convenient to express the energy loss in terms

View Verification Case 7 Model

[Verification Case 7](#)



Verification Case 8

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify8.fth

REFERENCE: Ernest Brater, Horace Williams, James Lindell, C.Y. Wei, Handbook of Hydraulics, 7th Ed., 1996, McGraw-Hill, Page 6.40-42, Example 6.3a

FLUID: Water at 21 deg. C

ASSUMPTIONS: N/A

RESULTS:

Parameter	Handbook	AFT Fathom
Head loss (meters)	41.99	42.14

DISCUSSION:

The exit loss factor of 1.0 is modeled as part of the exit Assigned Flow.

The overall head loss in AFT Fathom is obtained by subtracting the discharge EGL (17.86 meters) from the supply (60 meters). It is also displayed in the General Results list at the top of the Output window. Finally, the head loss is given in the Junction Deltas table at the top of the Output window. The junction delta was setup in the Output Control window.

The difference in the final answer (61.48 vs. 61.33) is due to round-off errors in the handbook calculations.

[List of All Verification Models](#)

Verification Case 8 Problem Statement

[Verification Case 8](#)

Ernest Brater, Horace Williams, James Lindell, C.Y. Wei, Handbook of Hydraulics, 7th Ed., 1996, McGraw-Hill, Page 6.40-42, Example 6.3a

[Brater, Williams, Lindell and Wei Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

6.40

HANDBOOK OF HYDRAULICS

A curve of reduction factors for bends of less than 90° given by the U.S. Bureau of Reclamation shows that the 90° bend loss should be multiplied by 0.83 for 60° bends, by 0.70 for 45° bends, and by 0.42 for 22.5° bends.

The solution of a pipe problem involving minor losses is illustrated by the following example.

Example 6.3. In Fig. 6.1 pipes *a* and *b* are steel pipes having diameters of 15 and 30 cm, respectively. The discharge is 0.10 m³/s, and the fluid is water at 21°C. The radii of the bends are twice the respective pipe diameters. Pipe lengths are

2-3	60 m
3-4	75 m
4-5	90 m
5-6	90 m

1. Determine the difference in elevations of the water surfaces in the two reservoirs ($z_1 - z_7$).

$$V_a = 0.10/0.0177 = 5.65 \text{ m/s}$$

$$V_b = 0.10/0.071 = 1.41 \text{ m/s}$$

Then

$$\frac{V_a^2}{2g} = 1.63 \text{ m} \quad \text{and} \quad \frac{V_b^2}{2g} = 0.10 \text{ m}$$

The various elements in the total energy loss are shown in Eq. (6.3) and identified below Eq. (6.3). They will now be evaluated.

From Eq. (6.34),

$$h_2 = 0.5 \frac{V_a^2}{2g} = 0.82 \text{ m}$$

From the section on turbulent flow,

$$h_{2-3} = f \frac{l}{d} \frac{V^2}{2g} = f \frac{60}{0.15} \times 1.63$$

The value of viscosity is obtained from Table 1.2. Then

PIPES

6.41

$$R_a = \frac{dV}{\nu} = \frac{0.15 \times 5.65}{0.984 \times 10^{-6}} = 0.86 \times 10^6$$

From Table 6.1,

$$\epsilon = 0.00005$$

and therefore

$$\frac{\epsilon}{d} = \frac{0.00005}{0.15} = 0.0003$$

From Fig. 6.4, $f = 0.016$. Then, from the preceding,

$$h_{2-3} = 0.016 \times 60/0.15 \times 1.63 = 10.43 \text{ m}$$

and, by similarity,

$$h_{3-4} = 13.04 \text{ m} \quad \text{and} \quad h_{4-5} = 15.65 \text{ m}$$

From Fig. 6.5,

$$h_3 = h_4 = 0.2 \frac{V_a^2}{2g} = 0.33 \text{ m}$$

From Table 6.5,

$$h_5 = 0.5 \times 1.63 = 0.82 \text{ m}$$

The Reynolds number for pipe b is

$$R_b = \frac{dV}{\nu} = \frac{0.30 \times 1.41}{0.984 \times 10^{-6}} = 0.43 \times 10^6$$

and from Fig. 6.4, $f = 0.0156$ and

$$h_{5-6} = 0.0156 \times 90/0.30 \times 0.10 = 0.47 \text{ m}$$

The outlet loss from Table 6.5 is

$$h_6 = \frac{V_b^2}{2g} = 0.10 \text{ m}$$

Then, as shown by Eq. (6.2), the difference in the elevation of the water surfaces can be found by summation of all the losses as follows:

6.42

HANDBOOK OF HYDRAULICS

$$\begin{aligned} z_1 - z_7 &= 0.82 + 10.43 + 0.33 + 13.04 + 0.33 \\ &\quad + 15.65 + 0.82 + 0.47 + 0.10 \\ &= 41.99 \text{ m} \end{aligned}$$

It should be noted that several of the so-called minor losses are greater than the friction loss in pipe *b*.

2. Find the pressure at *x* if the distance from point 4 to *x* is 50 m, $z_1 = 60$ m, and $z_x = 15$ m.

The solution of this problem is discussed under Fundamental Principles. The summation of the energy losses shown in Eq. (6.7) can be evaluated from the values in part 1 of this example, except for h_{4-x} , which is

$$h_{4-x} = 0.016 \times 50/0.15 \times 1.63 = 8.69 \text{ m}$$

Then

$$\Sigma h_l = 0.82 + 10.43 + 0.33 + 13.04 + 0.33 + 8.69 = 33.64 \text{ m}$$

Then, from Eq. (6.6),

$$\begin{aligned} \frac{p_x}{w} &= z_1 - \left(\frac{V_a^2}{2g} + z_x + \Sigma h_l \right) \\ &= 60 - (1.63 + 15 + 33.64) = 9.73 \text{ m} \end{aligned}$$

and

$$p_x = 9.8 \times 9.73 = 95.4 \text{ kN/m}^2$$

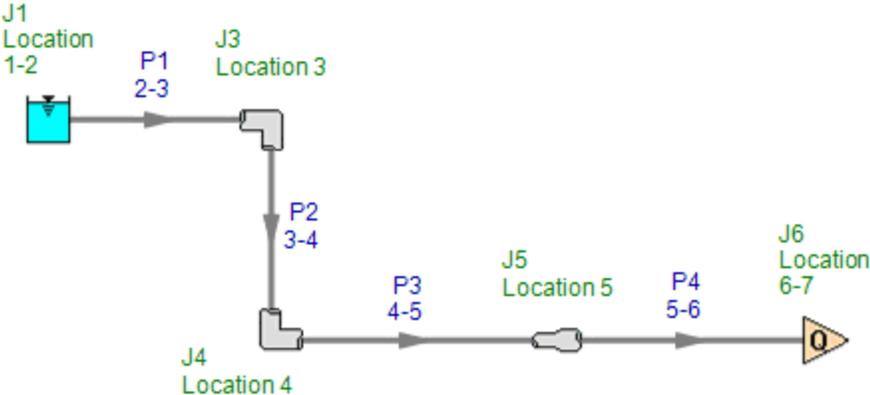
COMPOUND PIPES

Two types of compound-pipe problems may be encountered. One type occurs in pipe networks when two or more paths are available for water flowing between two points, such as points 1 and 5 in Fig. 6.6. A second type, illustrated by Fig. 6.7, occurs when water can flow to or from a junction of three or more pipes from independent sources or outlets.

The first type, often called the *parallel-pipe problem*, can be solved by making use of the fact that the total energy loss through the sys-

View Verification Case 8 Model

[Verification Case 8](#)



Verification Case 9

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify9.fth

REFERENCE: Ernest Brater, Horace Williams, James Lindell, C.Y. Wei, Handbook of Hydraulics, 7th Ed., 1996, McGraw-Hill, Page 6.40-42, Example 6.3b

FLUID: Water at 21 deg. C

ASSUMPTIONS: N/A

RESULTS:

Parameter	Handbook	AFT Fathom
Pressure at location x (kPa gauge)	95.4	94.13

DISCUSSION:

From the equation in the handbook, it is evident the pressure at location x is the static pressure and that it is gauge pressure. The units used, kN/m² are the same as kPa. The proper pressure to review in AFT Fathom for comparison to the handbook is the exit from junction 6.

The difference in the final answer is due to round-off errors in the handbook calculations.

[List of All Verification Models](#)

Verification Case 9 Problem Statement

[Verification Case 9](#)

Ernest Brater, Horace Williams, James Lindell, C.Y. Wei, Handbook of Hydraulics, 7th Ed., 1996, McGraw-Hill, Page 6.40-42, Example 6.3b

[Brater, Williams, Lindell and Wei Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

6.40

HANDBOOK OF HYDRAULICS

A curve of reduction factors for bends of less than 90° given by the U.S. Bureau of Reclamation shows that the 90° bend loss should be multiplied by 0.83 for 60° bends, by 0.70 for 45° bends, and by 0.42 for 22.5° bends.

The solution of a pipe problem involving minor losses is illustrated by the following example.

Example 6.3. In Fig. 6.1 pipes *a* and *b* are steel pipes having diameters of 15 and 30 cm, respectively. The discharge is 0.10 m³/s, and the fluid is water at 21°C. The radii of the bends are twice the respective pipe diameters. Pipe lengths are

2-3	60 m
3-4	75 m
4-5	90 m
5-6	90 m

1. Determine the difference in elevations of the water surfaces in the two reservoirs ($z_1 - z_7$).

$$V_a = 0.10/0.0177 = 5.65 \text{ m/s}$$

$$V_b = 0.10/0.071 = 1.41 \text{ m/s}$$

Then

$$\frac{V_a^2}{2g} = 1.63 \text{ m} \quad \text{and} \quad \frac{V_b^2}{2g} = 0.10 \text{ m}$$

The various elements in the total energy loss are shown in Eq. (6.3) and identified below Eq. (6.3). They will now be evaluated.

From Eq. (6.34),

$$h_2 = 0.5 \frac{V_a^2}{2g} = 0.82 \text{ m}$$

From the section on turbulent flow,

$$h_{2-3} = f \frac{l}{d} \frac{V^2}{2g} = f \frac{60}{0.15} \times 1.63$$

The value of viscosity is obtained from Table 1.2. Then

PIPES

6.41

$$R_a = \frac{dV}{\nu} = \frac{0.15 \times 5.65}{0.984 \times 10^{-6}} = 0.86 \times 10^6$$

From Table 6.1,

$$\epsilon = 0.00005$$

and therefore

$$\frac{\epsilon}{d} = \frac{0.00005}{0.15} = 0.0003$$

From Fig. 6.4, $f = 0.016$. Then, from the preceding,

$$h_{2-3} = 0.016 \times 60/0.15 \times 1.63 = 10.43 \text{ m}$$

and, by similarity,

$$h_{3-4} = 13.04 \text{ m} \quad \text{and} \quad h_{4-5} = 15.65 \text{ m}$$

From Fig. 6.5,

$$h_3 = h_4 = 0.2 \frac{V_a^2}{2g} = 0.33 \text{ m}$$

From Table 6.5,

$$h_5 = 0.5 \times 1.63 = 0.82 \text{ m}$$

The Reynolds number for pipe b is

$$R_b = \frac{dV}{\nu} = \frac{0.30 \times 1.41}{0.984 \times 10^{-6}} = 0.43 \times 10^6$$

and from Fig. 6.4, $f = 0.0156$ and

$$h_{5-6} = 0.0156 \times 90/0.30 \times 0.10 = 0.47 \text{ m}$$

The outlet loss from Table 6.5 is

$$h_6 = \frac{V_b^2}{2g} = 0.10 \text{ m}$$

Then, as shown by Eq. (6.2), the difference in the elevation of the water surfaces can be found by summation of all the losses as follows:

6.42

HANDBOOK OF HYDRAULICS

$$\begin{aligned} z_1 - z_7 &= 0.82 + 10.43 + 0.33 + 13.04 + 0.33 \\ &\quad + 15.65 + 0.82 + 0.47 + 0.10 \\ &= 41.99 \text{ m} \end{aligned}$$

It should be noted that several of the so-called minor losses are greater than the friction loss in pipe *b*.

2. Find the pressure at *x* if the distance from point 4 to *x* is 50 m, $z_1 = 60$ m, and $z_x = 15$ m.

The solution of this problem is discussed under Fundamental Principles. The summation of the energy losses shown in Eq. (6.7) can be evaluated from the values in part 1 of this example, except for h_{4-x} , which is

$$h_{4-x} = 0.016 \times 50/0.15 \times 1.63 = 8.69 \text{ m}$$

Then

$$\Sigma h_l = 0.82 + 10.43 + 0.33 + 13.04 + 0.33 + 8.69 = 33.64 \text{ m}$$

Then, from Eq. (6.6),

$$\begin{aligned} \frac{p_x}{w} &= z_1 - \left(\frac{V_a^2}{2g} + z_x + \Sigma h_l \right) \\ &= 60 - (1.63 + 15 + 33.64) = 9.73 \text{ m} \end{aligned}$$

and

$$p_x = 9.8 \times 9.73 = 95.4 \text{ kN/m}^2$$

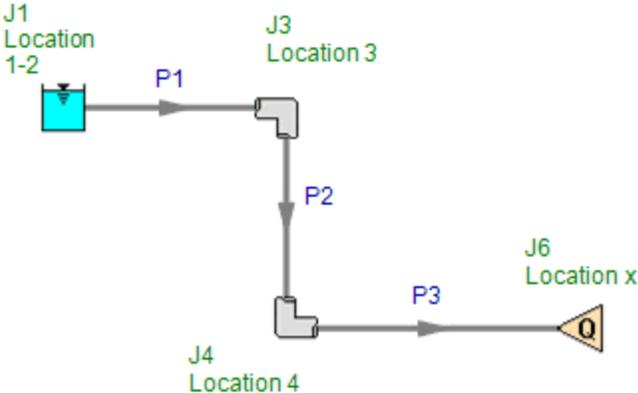
COMPOUND PIPES

Two types of compound-pipe problems may be encountered. One type occurs in pipe networks when two or more paths are available for water flowing between two points, such as points 1 and 5 in Fig. 6.6. A second type, illustrated by Fig. 6.7, occurs when water can flow to or from a junction of three or more pipes from independent sources or outlets.

The first type, often called the *parallel-pipe problem*, can be solved by making use of the fact that the total energy loss through the sys-

View Verification Case 9 Model

[Verification Case 9](#)



Verification Case 10

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify10.fth

REFERENCE: Ernest Brater, Horace Williams, James Lindell, C.Y. Wei, Handbook of Hydraulics, 7th Ed., 1996, McGraw-Hill, Page 6.47-48, Example 6.4

FLUID: Water

ASSUMPTIONS: Temperature unspecified, so assume 20 deg. C

RESULTS:

Pipe Flow Rate (m3/s)	1	2	3	4	5	6
Handbook	2.8	2.2	0.7	3.2	3.2	1.8
AFT Fathom	2.8	2.2	0.4	3.2	3.2	1.8

Node EGL (m)	1	2	3	4	5
Handbook	10.98	10	9.37	10.04	8.32
AFT Fathom	10.96	10	9.37	10.02	8.32

DISCUSSION:

The Handbook of Hydraulics solves the network using a different method than AFT Fathom, and also uses slightly different friction laws. The head loss relationship in the handbook assumes that the head loss in each pipe varies as the flow rate to the 1.85 power. AFT Fathom assumes it varies by the 2nd power. This slight difference accounts for the slightly different results.

Results for AFT Fathom vary somewhat from previous versions of AFT Fathom (prior to version 7) because the equation used to convert the Hazen-Williams factor to the Darcy-Weisbach friction factor was modified to use the traditional formula, as given in the AFT Fathom Help File.

[List of All Verification Models](#)

Verification Case 10 Problem Statement

Verification Case 10

Ernest Brater, Horace Williams, James Lindell, C.Y. Wei, Handbook of Hydraulics, 7th Ed., 1996, McGraw-Hill, Page 6.47-48, Example 6.4

Brater, Williams, Lindell and Wei Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

PIPES

6.47

- b. Find the algebraic sum of the head loss around the loop, after establishing a sign convention.
 - c. Compute the sum of the quantities nkQ_1^{n-1} from each pipe in the loop without regard to the direction.
 - d. Calculate the flow adjustment for the loop from Eq. (6.55).
4. Apply step 3 to each circuit in the system, make the flow adjustments to each pipe, and repeat the process until the desired accuracy is obtained.

Example 6.4. Determine the distribution of flow in the network of Fig. 6.8 with the inflow and outflow shown. Also, determine the elevation of the hydraulic grade line at points 1 and 5 if the reservoir elevation is 10 m. There is no flow into or out of the reservoir.

This network contains six pipes and two loops. The Hazen-Williams formula is used, and the coefficient is assumed to be 100 in each pipe. The lengths, diameters, and values of k for each pipe are listed in the following table:

Pipe	Length, m	Diameter, m	k	Assumed Q	ΔQ	Corrected Q
1	2000	2.0	0.146	-2.5	-0.3	-2.8
2	3000	2.0	0.218	+2.5	-0.3	+2.2
3	1000	2.0	0.073	+1.0	-0.3	+0.7
				-1.0	+0.3	-0.7
4	1000	2.0	0.073	-3.5	+0.3	-3.2
5	1000	1.8	0.122	-3.5	+0.3	-3.2
6	2000	1.5	0.590	+1.5	+0.3	+1.8

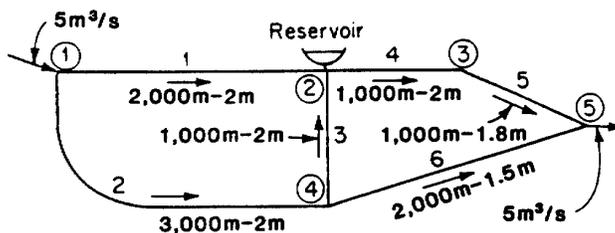


FIGURE 6.8 Distribution network.

The exponent in Eq. (6.49) is $n = 1.85$. Flows in the counterclockwise direction were taken as positive. The assumed flows with the corresponding signs for the flow directions shown in Fig. 6.8 are also listed in the table. Note that the discharge in pipe 3 has a plus sign for the left-hand loop and a minus sign for the right-hand loop. Flow directions are computed for the left and right loops in accordance with steps 3a to 3d in the procedure outlined. The computed correction factors and the corrected discharges are also listed. The correction factors for both loops are applied to pipe 3, and both decrease the discharge.

The process is illustrated in the following table. It should be repeated until the pipe flows do not change significantly.

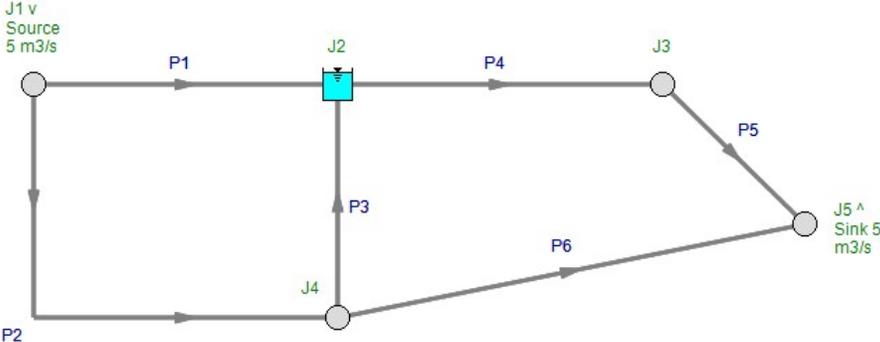
Pipe	ΣkQ^n	ΣnkQ^{n-1}
Left loop		
1	$0.146 \times 2.5^{1.85} = -0.80$	$1.85 \times 0.146 \times 2.5^{0.85} = 0.59$
2	$0.218 \times 2.5^{1.85} = +1.19$	$1.85 \times 0.218 \times 2.5^{0.85} = 0.88$
3	$0.073 \times 1.0^{1.85} = +0.073$	$1.85 \times 0.073 \times 1.0^{0.85} = 0.14$
		1.61
	$\Delta Q = \frac{-(-0.80 + 1.19 + 0.073)}{1.61} = -0.3$	
Right loop		
3	$0.073 \times 1.0^{1.85} = -0.073$	$1.85 \times 0.073 \times 1.0^{0.85} = 0.14$
4	$0.073 \times 3.5^{1.85} = -0.74$	$1.85 \times 0.073 \times 3.5^{0.85} = 0.39$
5	$0.122 \times 3.5^{1.85} = -1.24$	$1.85 \times 0.122 \times 3.5^{0.85} = 0.65$
6	$0.59 \times 1.5^{1.85} = +1.25$	$1.85 \times 0.59 \times 1.5^{0.85} = 1.54$
		2.72
	$\Delta Q = \frac{-(-0.073 - 0.74 - 1.24 + 1.25)}{2.72} = +0.3$	

In this example the flows are balanced after the first adjustment since the originally assumed flows were quite close.

The pressure level at junction 3 is $10.0 - 0.073 \times 3.2^{1.85} = 9.37$ m, and at the outflow junction it is $9.37 - 0.122 \times 3.2^{1.85} = 8.32$ m. At node 1 it is $10.0 + 0.146 \times 2.8^{1.85} = 10.98$ m.

View Verification Case 10 Model

[Verification Case 10](#)



Verification Case 11

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify11.fth

REFERENCE: Ernest Brater, Horace Williams, James Lindell, C.Y. Wei, Handbook of Hydraulics, 7th Ed., 1996, McGraw-Hill, Page 13.14, Example 13.9, results on page 13.39

FLUID: Water

ASSUMPTIONS: Temperature unspecified, so assume 20 deg. C

RESULTS:

Pipe Flow Rate (m3/s)	1	2	3	4	5	6
Handbook	3.42	2.68	0.89	3.21	3.21	1.79
AFT Fathom	3.48	2.73	0.94	3.21	3.21	1.79

Node EGL (m)	1	2	3	4	5
Handbook	11.42	10.00	9.37	10.06	8.32
AFT Fathom	11.46	10.00	9.38	10.06	8.33

DISCUSSION:

The Handbook of Hydraulics solves the network using a different method than AFT Fathom, and also uses slightly different friction laws. The head loss relationship in the handbook assumes that the head loss in each pipe varies as the flow rate to the 1.85 power. AFT Fathom assumes it varies by the 2nd power. This slight difference accounts for the slightly different results.

Results for AFT Fathom vary somewhat from previous versions of AFT Fathom (prior to version 7) because the equation used to convert the Hazen-Williams factor to the Darcy-Weisbach friction factor was modified to use the traditional formula, as given in the AFT Fathom Help File.

[List of All Verification Models](#)

Verification Case 11 Problem Statement

[Verification Case 11](#)

Ernest Brater, Horace Williams, James Lindell, C.Y. Wei, Handbook of Hydraulics, 7th Ed., 1996, McGraw-Hill, Page 13.14, Example 13.9, results on page 13.39

[Brater, Williams, Lindell and Wei Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

13.14

HANDBOOK OF HYDRAULICS

Nodal Method). The following example illustrates the development of a computer program to solve the pipe network problems used as examples in Sec. 6.

Example 13.9. Determine the flow distribution in the network presented in Example 6.5 by use of the nodal method. A schematic diagram of the system is shown in Fig. 6.8. Note that a pump, with characteristics shown in Fig. 6.9, is located at node 1, and the flow at the reservoir (junction 2) is determined from the analysis.

The program listing and input data are shown in the Appendix. One of the primary difficulties associated with a computer solution of a complex network is the adequate description of the physical and geometric properties of the system. In this program the network is described by assigning a nominal upstream and downstream node to each pipe. Each junction is assigned a number indicating the type of junction: 1—node with one or more pipes connected, 2—node with a pump, 3—node at a reservoir, and 4—node with outflow.

For each pipe the upstream and downstream nodes are specified as well as pipe length, diameter, and Hazen-Williams roughness coefficient. For each junction the type is given along with a first guess of the hydraulic grade line and, depending on the type, the demand Q_v or the elevation on the suction side of the pump. If a pump is present, then the rating is input as a table of discharge versus head.

Execution of the program is stopped by exceeding the number of iterations given by `maxit`, by finding the sum of the junction head corrections, or by any iteration that is less than a tolerance `toler`.

The output for this example are shown in the Appendix. The first table shows the variation in hydraulic grade line with each iteration and the convergence tolerance. The second table shows the flow in each pipe.

For large complex pipe networks more advanced computer programs are available, as indicated in Sec. 14.

UNSTEADY FLOW IN PIPES AND OPEN CHANNELS

The method of characteristics can be used to solve the unsteady flow equations.⁸ The equations of motion and continuity are combined and then, using the method of characteristics, two new equations are derived,

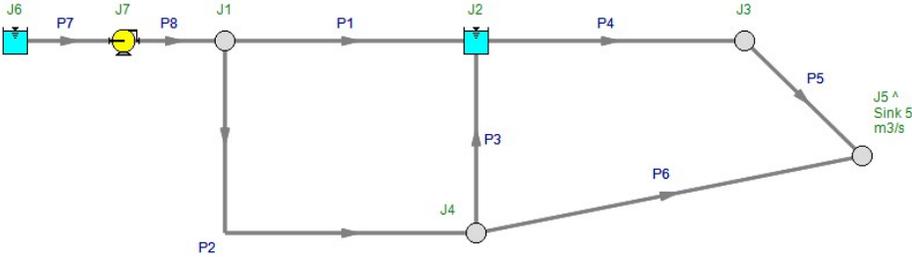
Output Data

Iteration	Tolerance	H(1..5)				
0	0.0000	11.00	10.00	9.00	10.50	8.50
1	1.8343	11.47	10.00	9.43	9.89	8.18
2	0.6517	11.39	10.00	9.32	10.20	8.33
3	0.2895	11.44	10.00	9.37	10.03	8.31
4	0.0642	11.41	10.00	9.37	10.06	8.31
5	0.0105	11.42	10.00	9.37	10.06	8.32
6	0.0026	11.42	10.00	9.37	10.06	8.32

Pipe	Flow
1	3.42
2	2.68
3	0.89
4	3.21
5	3.21
6	1.79

View Verification Case 11 Model

[Verification Case 11](#)



Verification Case 12

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify12.fth

REFERENCE: Ingersoll-Dresser Pumps, Cameron Hydraulic Data, 18th Ed., 1995, Published by Ingersoll-Dresser Pumps, Page 1-13, Example 1

FLUID: Water at 68 deg. F

ASSUMPTIONS: N/A

RESULTS:

Parameter	Cameron	AFT Fathom
NPSHA (feet)	20.26	20.23
Suction Head (feet)	-12.92	-12.92

DISCUSSION:

The example in the book does not actually perform a flow calculation. To demonstrate how AFT Fathom determines NPSHA, a few unusual steps were taken.

1. The head loss up the pump is given in Cameron as 2.92 feet. This was modeled in valve J2, as a fixed pressure drop of 2.92 feet.
2. All three pipes were modeled as frictionless so that all pressure loss comes from valve J2.
3. A discharge pipe and reservoir were modeled to complete the system, even though the discharge piping does not affect the answer to the problem.
4. The pump was modeled as an assigned flow at 100 gpm. Since the pressure drop is fixed at J2 as 2.92 feet, the flow rate in the pump (and piping for that matter) does not affect the answer. In typical system analysis, the flow in the piping affects the results because as the flow changes the pressure drop changes. In this model this did not occur because all pipes were frictionless.

The AFT Fathom NPSHA prediction is shown in the Pump Summary at the top of the Output window. The suction head is shown as the EGL inlet to the pump, J3. It is the negative of the suction lift.

[List of All Verification Models](#)

Verification Case 12 Problem Statement

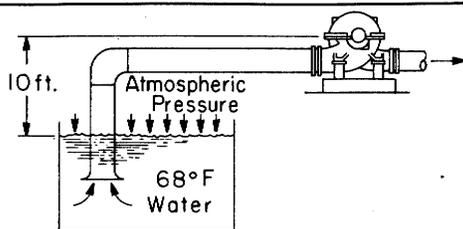
[Verification Case 12](#)

Ingersoll-Dresser Pumps, Cameron Hydraulic Data, 18th Ed., 1995, Published by Ingersoll-Dresser Pumps, Page 1-13, Example 1

[Cameron Hydraulic Data Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Hydraulics



1

Fig. 4. (Example No 1)

Example No 1 (Fig 4)

Open system, source below pump; 68°F water at sea level. Atmospheric pressures 14.696 psia, 33.96 ft abs. Vapor pressure of liquid 0.339 psia = 0.783 ft abs.

$$\text{NPSHA} = 33.96 - 0.783 - 10.00 - 2.92 = 20.26 \text{ ft}$$

Suction Lift = 10.00 + 2.92 = 12.92 ft—this is to be added to discharge head to obtain total head.

Note: No pump can actually lift water on the suction side. In this case, water is forced in by an excess of atmospheric pressure over the vapor pressure less 12.92 ft net static lift.

Example No 2 (Fig 5)

Open system, source above pump; 68°F water at sea level.

$$\text{NPSHA} = 33.96 - 0.783 + 10.00 - 2.92 = 40.26 \text{ ft.}$$

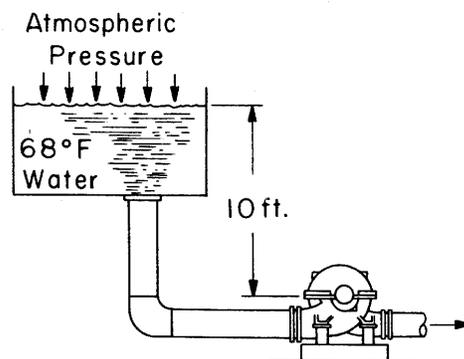
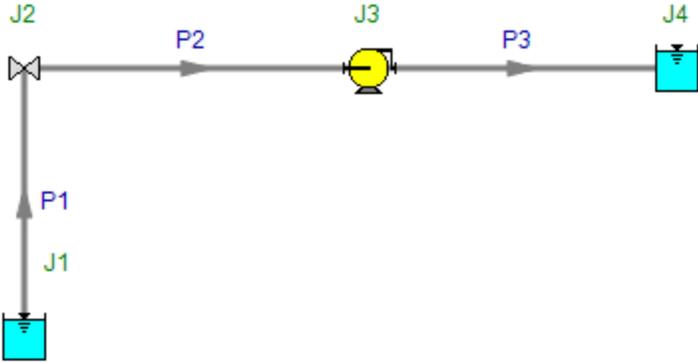


Fig. 5 (Example 2)

View Verification Case 12 Model

[Verification Case 12](#)



Verification Case 13

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify13.fth

REFERENCE: Ingersoll-Dresser Pumps, Cameron Hydraulic Data, 18th Ed., 1995, Published by Ingersoll-Dresser Pumps, Page 1-13, 14, Example 2

FLUID: Water at 68 deg. F

ASSUMPTIONS: N/A

RESULTS:

Parameter	Cameron	AFT Fathom
NPSHA (feet)	40.26	40.23
Suction Head (feet)	7.08	7.08

DISCUSSION:

The example in the book does not actually perform a flow calculation. To demonstrate how AFT Fathom determines NPSHA, a few unusual steps were taken.

1. The head loss up the pump is given in Cameron as 2.92 feet. This was modeled in the valve J2, as a fixed pressure drop of 2.92 feet.
2. All three pipes were modeled as frictionless so that all pressure loss comes from valve J2.
3. A discharge pipe and reservoir were modeled to complete the system, even though the discharge piping does not affect the answer to the problem.
4. The pump was modeled as an assigned flow at 100 gpm. Since the pressure drop is fixed at J2 as 2.92 feet, the flow rate in the pump (and piping for that matter) does not affect the answer. In typical system analysis, the flow in the piping affects the results because as the flow changes the pressure drop changes. In this model this did not occur because all pipes were frictionless.

The AFT Fathom NPSHA prediction is shown in the Pump Summary at the top of the Output window. The suction head is shown as the EGL inlet to the pump, J3.

[List of All Verification Models](#)

Verification Case 13 Problem Statement

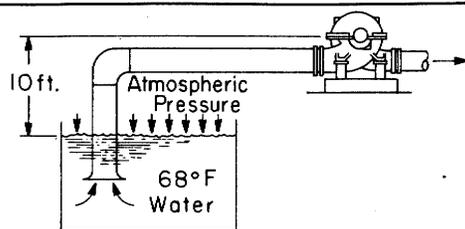
[Verification Case 13](#)

Ingersoll-Dresser Pumps, Cameron Hydraulic Data, 18th Ed., 1995, Published by Ingersoll-Dresser Pumps, Page 1-13, 14, Example 2

[Cameron Hydraulic Data Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Hydraulics



1

Fig. 4. (Example No 1)

Example No 1 (Fig 4)

Open system, source below pump; 68°F water at sea level. Atmospheric pressures 14.696 psia, 33.96 ft abs. Vapor pressure of liquid 0.339 psia = 0.783 ft abs.

$$\text{NPSHA} = 33.96 - 0.783 - 10.00 - 2.92 = 20.26 \text{ ft}$$

Suction Lift = 10.00 + 2.92 = 12.92 ft—this is to be added to discharge head to obtain total head.

Note: No pump can actually lift water on the suction side. In this case, water is forced in by an excess of atmospheric pressure over the vapor pressure less 12.92 ft net static lift.

Example No 2 (Fig 5)

Open system, source above pump; 68°F water at sea level.

$$\text{NPSHA} = 33.96 - 0.783 + 10.00 - 2.92 = 40.26 \text{ ft.}$$

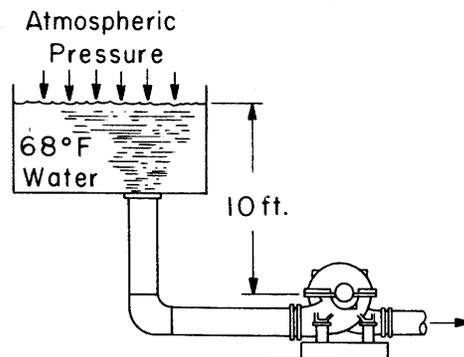


Fig. 5 (Example 2)

I Ingersoll-Dresser Pumps Cameron Hydraulic Data

Suction Head— $10.00 - 2.92 = 7.08$ ft—this is to be subtracted from discharge head to obtain total head.

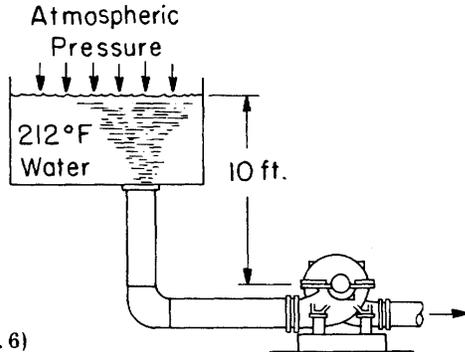


Fig. 6 (Example 3)

Example No. 3 (Fig. 6)

Open system, source above pump; 212°F water at sea level; vapor pressure same as atmospheric since liquid at boiling point. At 212°F, the vapor pressure of water—in feet of 212°F water—is 35.38 ft. ($14.696 \times 144 \div 59.81 = 35.38$).

$NPSHA = 35.38 - 35.38 + 10.00 - 2.92 = 7.08$ ft. In this case, atmospheric pressure does not add to NPSHA since it is required to keep the water in liquid phase.

Suction Head = $10.00 - 2.92 = 7.08$ ft—this is to be subtracted from discharge head to obtain total head.

Note: In this example it was assumed that pipe friction losses for 212°F water were the same as for 68°F water whereas actually they would be somewhat less, as will also be the case in Example 4.

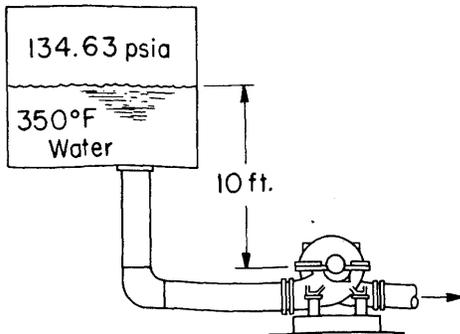
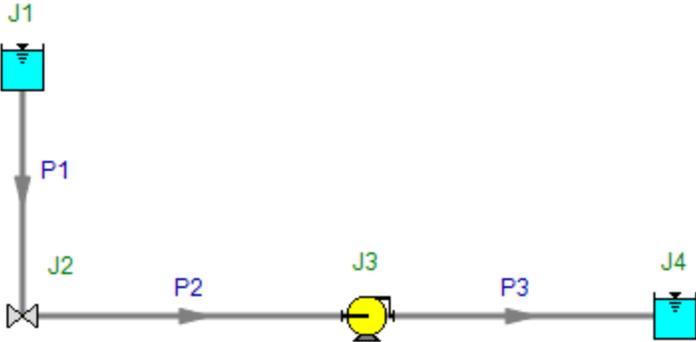


Fig. 7 (Example 4)

View Verification Case 13 Model

[Verification Case 13](#)



Verification Case 14

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify14.fth

REFERENCE: Ingersoll-Dresser Pumps, Cameron Hydraulic Data, 18th Ed., 1995, Published by Ingersoll-Dresser Pumps, Page 1-14, Example 3

FLUID: Water at 212 deg. F, 1 atm

ASSUMPTIONS: N/A

RESULTS:

Parameter	Cameron	AFT Fathom
NPSHA (feet)	7.08	7.048
Suction Head (feet)	7.08	7.080

DISCUSSION:

The example in the book does not actually perform a flow calculation. To demonstrate how AFT Fathom determines NPSHA, a few unusual steps were taken.

1. The head loss up the pump is given in Cameron as 2.92 feet. This was modeled in the valve J2, as a fixed pressure drop of 2.92 feet.
2. All three pipes were modeled as frictionless so that all pressure loss comes from valve J2.
3. A discharge pipe and reservoir were modeled to complete the system, even though the discharge piping does not affect the answer to the problem.
4. The pump was modeled as an assigned flow at 100 gpm. Since the pressure drop is fixed at J2 as 2.92 feet, the flow rate in the pump (and piping for that matter) does not affect the answer. In typical system analysis, the flow in the piping affects the results because as the flow changes the pressure drop changes. In this model this did not occur because all pipes were frictionless.

The AFT Fathom NPSHA prediction is shown in the Pump Summary at the top of the Output window. The suction head is shown as the EGL inlet to the pump, J3.

[List of All Verification Models](#)

Verification Case 14 Problem Statement

Verification Case 14

Ingersoll-Dresser Pumps, Cameron Hydraulic Data, 18th Ed., 1995, Published by Ingersoll-Dresser Pumps, Page 1-14, Example 3

[Cameron Hydraulic Data Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

ID Ingersoll-Dresser Pumps Cameron Hydraulic Data

Suction Head— $10.00 - 2.92 = 7.08$ ft—this is to be subtracted from discharge head to obtain total head.

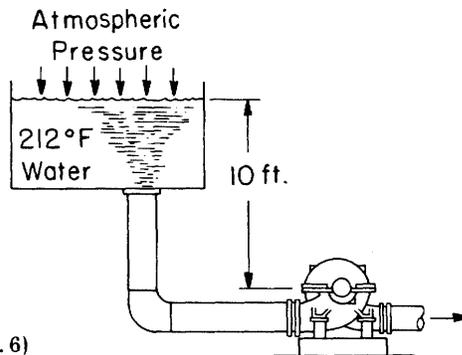


Fig. 6 (Example 3)

Example No. 3 (Fig. 6)

Open system, source above pump; 212°F water at sea level; vapor pressure same as atmospheric since liquid at boiling point. At 212°F, the vapor pressure of water—in feet of 212°F water—is 35.38 ft. ($14.696 \times 144 \div 59.81 = 35.38$).

$NPSHA = 35.38 - 35.38 + 10.00 - 2.92 = 7.08$ ft. In this case, atmospheric pressure does not add to NPSHA since it is required to keep the water in liquid phase.

Suction Head = $10.00 - 2.92 = 7.08$ ft—this is to be subtracted from discharge head to obtain total head.

Note: In this example it was assumed that pipe friction losses for 212°F water were the same as for 68°F water whereas actually they would be somewhat less, as will also be the case in Example 4.

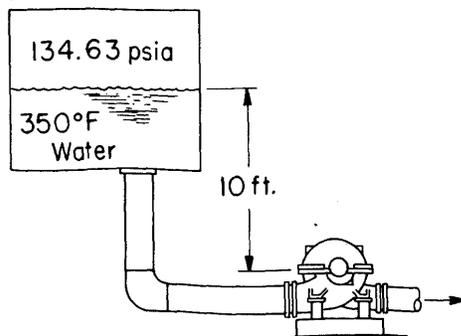
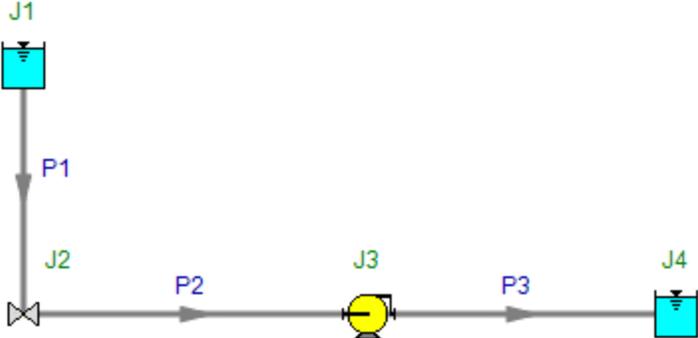


Fig. 7 (Example 4)

View Verification Case 14 Model

[Verification Case 14](#)



Verification Case 15

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify15.fth

REFERENCE: Ingersoll-Dresser Pumps, Cameron Hydraulic Data, AF 18th Ed., 1995, Published by Ingersoll-Dresser Pumps, Page 1-15, Example 4

FLUID: Water at 350 deg. F, 134.63 psia

ASSUMPTIONS: N/A

RESULTS:

Parameter	Cameron	AFT Fathom
NPSHA (feet)	7.08	7.158
Suction Head (feet)	317.69	317.71

DISCUSSION:

The example in the book does not actually perform a flow calculation. To demonstrate how AFT Fathom determines NPSHA, a few unusual steps were taken.

1. The head loss up the pump is given in Cameron as 2.92 feet. This was modeled in the valve J2, as a fixed pressure drop of 2.92 feet.
2. All three pipes were modeled as frictionless so that all pressure loss comes from valve J2.
3. A discharge pipe and reservoir were modeled to complete the system, even though the discharge piping does not affect the answer to the problem.
4. The pump was modeled as an assigned flow at 100 gpm. Since the pressure drop is fixed at J2 as 2.92 feet, the flow rate in the pump (and piping for that matter) does not affect the answer. In typical system analysis, the flow in the piping affects the results because as the flow changes the pressure drop changes. In this model this did not occur because all pipes were frictionless.

The AFT Fathom NPSHA prediction is shown in the Pump Summary at the top of the Output window. The suction head is shown as the EGL inlet to the pump, J3.

[List of All Verification Models](#)

Verification Case 15 Problem Statement

[Verification Case 15](#)

Ingersoll-Dresser Pumps, Cameron Hydraulic Data, 18th Ed., 1995, Published by Ingersoll-Dresser Pumps, Page 1-15, Example 4

[Cameron Hydraulic Data Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Hydraulics

Example No 4 (Fig 7)

Closed system (under pressure as a feed water deaerator) source above pump.

350°F water V.P. = 134.60 psia = 348.67 ft abs (at 350°F sp gr = 0.8904).

1

$$\text{NPSHA} = 348.67 - 348.67 + 10.00 - 2.92 = 7.08 \text{ ft.}$$

Suction Head—(Figure basis gage pressures; i.e., 119.91 psig = 310.61 ft) = 310.61 + 10.00 - 2.92 = 317.69 ft—This is to be subtracted from the discharge head to obtain total system head. It is important to note that while the suction head is 317.69 ft (122.67 psig) the NPSHA is still only 7.08 ft.

Example No 5 (Fig 8)—Closed system (under vacuum as a condenser hotwell) liquid source above pump. Absolute pressure (h_a) = 1.50" Hg \times 1.1349 = 1.70 ft. Water at saturation point 91.72°F; therefore vapor pressure (h_{vpa}) = 1.50" Hg \times 1.1349 = 1.70 ft.

$$\text{NPSHA} = 1.70 - 1.70 + 10.00 - 2.92 = 7.08 \text{ ft.}$$

Suction Condition—In this example the suction condition (head or lift) for the pump can best be visualized by the calculations listed below where it can be seen that we have a suction lift equal to the vacuum effect at the suction source less the net static submergence.

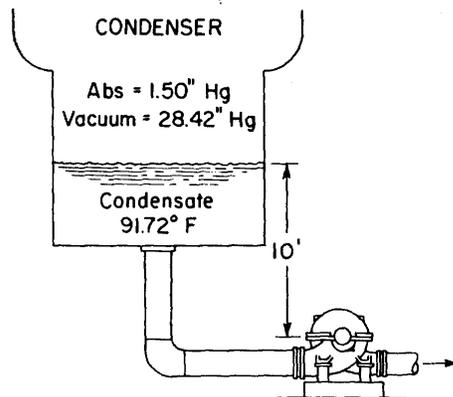
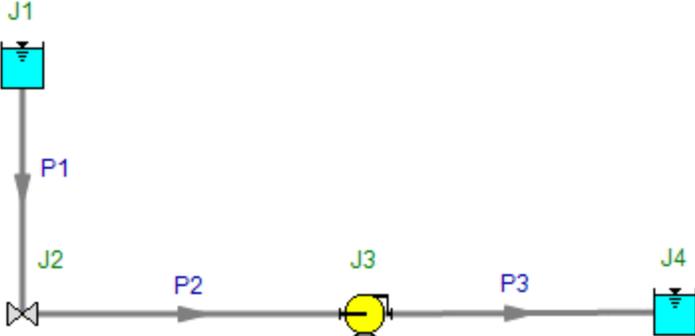


Fig. 8 (Example 5)

View Verification Case 15 Model

[Verification Case 15](#)



Verification Case 16

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify16.fth

REFERENCE: Ingersoll-Dresser Pumps, Cameron Hydraulic Data, 18th Ed., 1995, Published by Ingersoll-Dresser Pumps, Page 1-15, 16, Example 5

FLUID: Water at 91.72 deg. F, 1.5 in. Hg

ASSUMPTIONS: N/A

RESULTS:

Parameter	Cameron	AFT Fathom
NPSHA (feet)	7.08	7.08
Suction Head (feet)	-25.17	-25.30

DISCUSSION:

The example in the book does not actually perform a flow calculation. To demonstrate how AFT Fathom determines NPSHA, a few unusual steps were taken.

1. The head loss up the pump is given in Cameron as 2.92 feet. This was modeled in the valve J2, as a fixed pressure drop of 2.92 feet.
2. All three pipes were modeled as frictionless so that all pressure loss comes from valve J2.
3. A discharge pipe and reservoir were modeled to complete the system, even though the discharge piping does not affect the answer to the problem.
4. The pump was modeled as an assigned flow at 100 gpm. Since the pressure drop is fixed at J2 as 2.92 feet, the flow rate in the pump (and piping for that matter) does not affect the answer. In typical system analysis, the flow in the piping affects the results because as the flow changes the pressure drop changes. In this model this did not occur because all pipes were frictionless.

The AFT Fathom NPSHA prediction is shown in the Pump Summary at the top of the Output window. The suction head is shown as the EGL inlet to the pump, J3. It is the negative of the suction lift.

[List of All Verification Models](#)

Verification Case 16 Problem Statement

[Verification Case 16](#)

Ingersoll-Dresser Pumps, Cameron Hydraulic Data, 18th Ed., 1995, Published by Ingersoll-Dresser Pumps, Page 1-15, 16, Example 5

[Cameron Hydraulic Data Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Hydraulics

Example No 4 (Fig 7)

Closed system (under pressure as a feed water deaerator) source above pump.

350°F water V.P. = 134.60 psia = 348.67 ft abs (at 350°F sp gr = 0.8904).

$$\text{NPSHA} = 348.67 - 348.67 + 10.00 - 2.92 = 7.08 \text{ ft.}$$

Suction Head—(Figure basis gage pressures; i.e., 119.91 psig = 310.61 ft) = 310.61 + 10.00 - 2.92 = 317.69 ft—This is to be subtracted from the discharge head to obtain total system head. It is important to note that while the suction head is 317.69 ft (122.67 psig) the NPSHA is still only 7.08 ft.

Example No 5 (Fig 8)—Closed system (under vacuum as a condenser hotwell) liquid source above pump. Absolute pressure (h_a) = 1.50" Hg \times 1.1349 = 1.70 ft. Water at saturation point 91.72°F; therefore vapor pressure (h_{vpa}) = 1.50" Hg \times 1.1349 = 1.70 ft.

$$\text{NPSHA} = 1.70 - 1.70 + 10.00 - 2.92 = 7.08 \text{ ft.}$$

Suction Condition—In this example the suction condition (head or lift) for the pump can best be visualized by the calculations listed below where it can be seen that we have a suction lift equal to the vacuum effect at the suction source less the net static submergence.

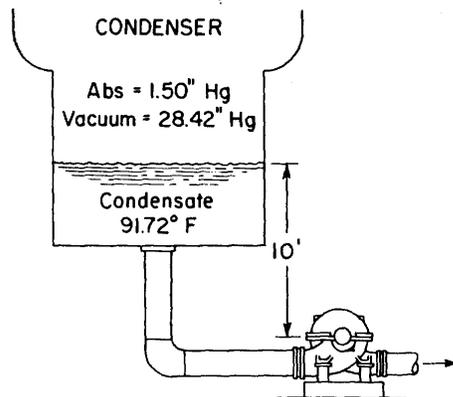


Fig. 8 (Example 5)

ID Ingersoll-Dresser Pumps Cameron Hydraulic Data

28.42" Hg Vacuum = 28.42 x 1.1349	=	32.25 ft
Static submergence		10.00 ft
Friction and entrance loss		<u>- 2.95 ft</u>
Net static submergence	=	7.08 ft <u>- 7.08 ft</u>
Equivalent suction lift = vacuum effect less net submergence	=	25.17 ft

In this example it is noted that the NPSHA is equal to the static suction head less the friction and entrance losses. Also the equivalent suction lift must be added to the total discharge head to obtain the total system head.

In the foregoing examples standard sea level atmospheric conditions were assumed; for other locations where altitude is a factor proper corrections must be made. These examples (3, 4 and 5) illustrate that if the liquid is in equilibrium (vapor pressure corresponds to saturation temperature) then the NPSH is equal to the difference in elevation between the liquid supply level and the pump centerline elevation (or impeller eye) less the sum of the entrance loss and the friction losses in the suction line.

NPSH reductions—hydrocarbon liquids and hot water

The NPSH requirements of centrifugal pumps are normally determined on the basis of handling water at or near normal room temperatures. However, field experience and laboratory tests have confirmed that pumps handling certain gas free hydrocarbon fluids and water at elevated temperatures will operate satisfactorily with harmless cavitation and less NPSH available than would be required for cold water.

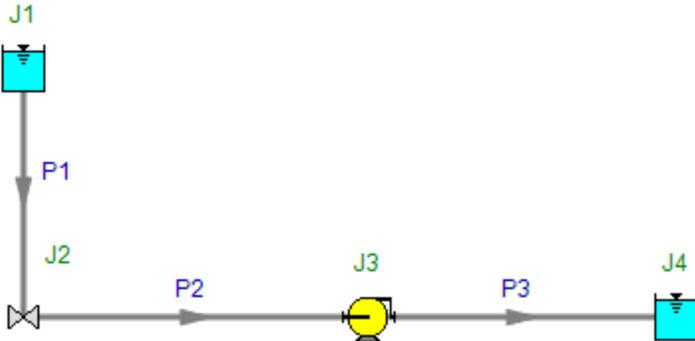
The figure on page 1-52 shows NPSH reductions that may be considered for hot water and certain gas free pure hydrocarbon liquids.

The use and application of this chart is subject to certain limitations some of which are summarized below:

1. The NPSH reductions shown are based on laboratory test data at steady state suction conditions and on the gas free pure hydrocarbon liquids shown; its application to other liquids must be considered experimental and is not recommended.
2. No NPSH reduction should exceed 50% of the NPSH required for cold water or ten feet whichever is smaller.

View Verification Case 16 Model

[Verification Case 16](#)



Verification Case 17

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify17.fth

REFERENCE: Ingersoll-Dresser Pumps, Cameron Hydraulic Data, 18th Ed., 1995, Published by Ingersoll-Dresser Pumps, Page 3-9, 10

FLUID: Water at 68 deg. F

ASSUMPTIONS: N/A

RESULTS:

Parameter	Cameron	AFT Fathom
Suction head (feet)	-5.35	-5.35
Discharge head (feet)	290	289.7
Pump head (feet)	295	295.0
NPSHA (feet)	Not provided	27.80

DISCUSSION:

The losses at the pump suction are not explicitly provided in AFT Fathom's library, so the total K factor of 1.57 for the foot valve and long radius elbow is entered as a fitting and loss value in pipe P1. The losses in the discharge piping are in AFT Fathom's library, and are selected as fitting and loss values in pipe P2. The only loss not shown in pipe P2 is the entrance loss at the discharge tank, and that is modeled in the reservoir junction, J3.

The 200 gpm is modeled at the pump as an assigned flow pump.

The AFT Fathom NPSHA prediction is shown in the Pump Summary at the top of the Output window. The suction head is shown as the difference between the EGL inlet and the elevation (23.27 – 28.62) at the pump, J2. It is the negative of the suction lift. The discharge head is shown as the difference between the EGL outlet and the elevation (318.30 – 28.62) at the pump.

[List of All Verification Models](#)

Verification Case 17 Problem Statement

[Verification Case 17](#)

Ingersoll-Dresser Pumps, Cameron Hydraulic Data, 18th Ed., 1995, Published by Ingersoll-Dresser Pumps, Page 3-9, 10

[Cameron Hydraulic Data Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Friction

Friction—head loss—sample calculation:

To illustrate the application of the friction and head loss data in calculating the total system head for a specific system the following example is offered:

Problem—referring to the accompanying figure, page 3-10, a pump takes water (68°F) from a sump and delivers it through 1250 feet of 4" diameter schedule 40 steel pipe. The suction pipe is 4" vertical 5 feet long and includes a foot valve and a long-radius elbow. The discharge line includes two standard 90 degree flanged elbows, a swing check valve and an open wedge—disc gate valve. It is required to find the suction lift (h_s) and the discharge head (h_d) when the rate of flow is 200 gpm.

3

Solution

(a) SUCTION LIFT—Data from table on page 3-20.

$$\text{Velocity head} = \frac{V^2}{2g} = 0.395 \text{ ft}$$

Pipe friction loss $h_f = 2.25$ ft per 100 ft of pipe.

The resistance coefficient for the foot valve (page 3-115) is $K = 1.3$ and for the long-radius elbow (page 3-112) is $K = 0.27$.

The head loss due to pipe friction will be:

$$h_f = 2.25 \times \frac{5}{100} = 0.11 \text{ ft}$$

The head loss in the foot valve and long-radius elbow will be:

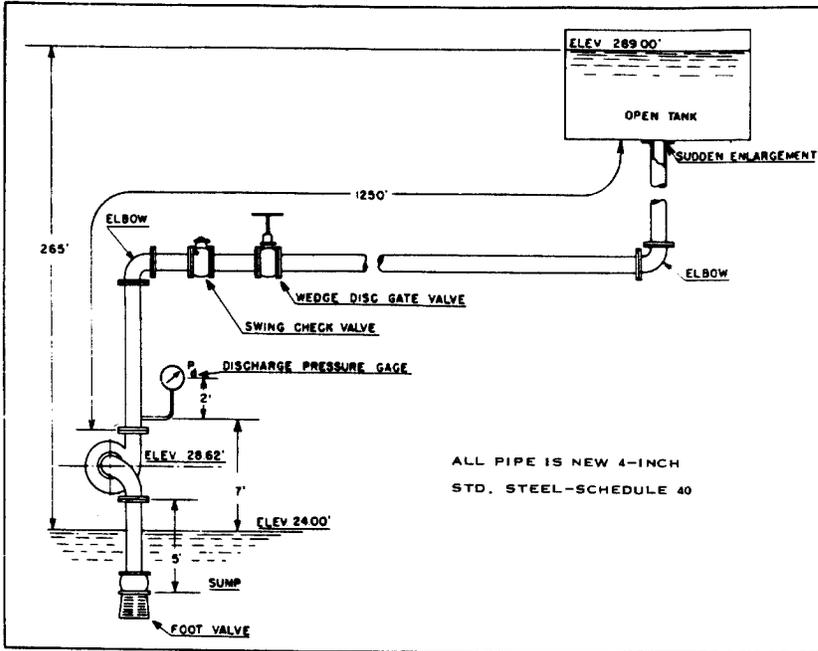
$$h_f = K \frac{V^2}{2g} = (1.3 + 0.27) \times 0.395 = 0.62 \text{ ft}$$

Total suction lift (h_s) = (28.62 - 24.00) + 0.62 + 0.11 = 5.35 ft

(b) DISCHARGE HEAD—The head loss due to pipe friction in the 4" discharge line will be:

$$h_f = 2.25 \times \frac{1250}{100} = 28.13 \text{ ft}$$

Ingersoll-Dresser Pumps Cameron Hydraulic Data



The resistance coefficient for the various fittings as obtained from the tables will be:

Standard 90 degree flanged elbow (pg. 3-112)	K = 0.51
Swing check valve (pg. 3-115)	K = 1.70
Wedge-disc gate valve (pg. 3-111)	K = 0.14
Sudden enlargement (pg. 3-116 to 3-118)	K = 1.00

The total resistance coefficient for the fittings on the discharge side and sudden enlargement at exit will be:

$$K = 2 \times 0.51 + 1.70 + 0.14 + 1.0 = 3.86$$

Therefore the head loss due to the fittings on the discharge side and sudden enlargement will be:

$$h_f = K \frac{V^2}{2g} = 3.86 \times 0.395 = 1.52 \text{ ft}$$

The total discharge head (h_d) will be:

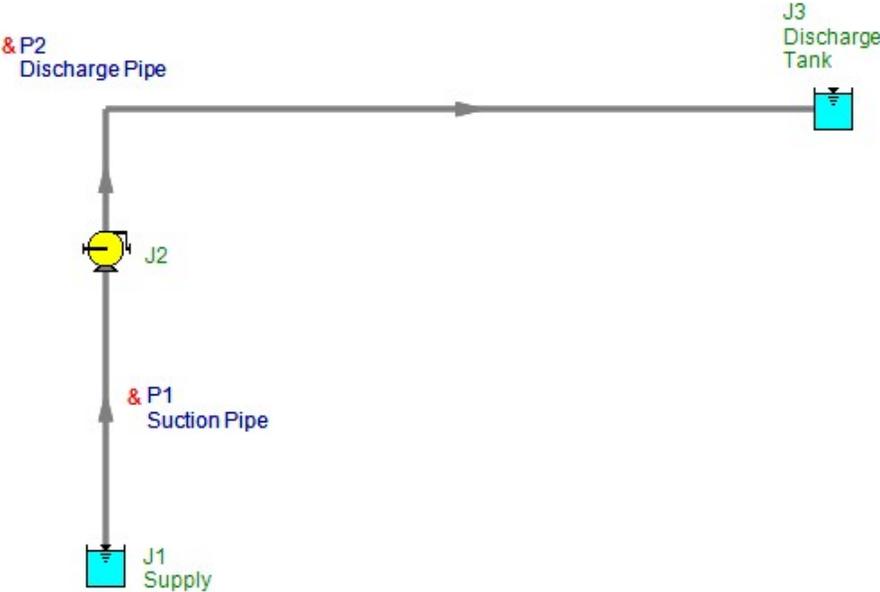
$$h_d = (289.00 - 28.62) + 1.52 + 28.13 = 290 \text{ ft}$$

$$\text{Total system head (H)} = h_d + h_s = 290 + 5.35 = 295 \text{ ft}$$

Add a reasonable safety factor to allow for any abnormal condition of pipe's interior or surface (see page 3-5).

View Verification Case 17 Model

[Verification Case 17](#)



Verification Case 18

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify18.fth

REFERENCE: Igor Karassik, William Krutzsch, Warren Fraser, Joseph Messina, Pump Handbook, 2nd Ed., 1986, McGraw-Hill, Page 8.9-11, Example 1

FLUID: Pumped system with fluid of 0.8 specific gravity

ASSUMPTIONS: N/A

RESULTS:

Parameter	Handbook	AFT Fathom
Pump head (feet)	372	371.6

DISCUSSION:

The suction and discharge pipe head loss of 3 and 25 feet, respectively, was modeled as a fitting and loss K factor to obtain the proper head loss. Both pipes were modeled as frictionless. Thus the pipe lengths, which were not specified, do not affect the head loss. In addition, the viscosity of the fluid, which was not specified, does not affect the head loss since the pipe are frictionless. Thus a value of 1 lbm/hr-ft was assumed.

The 1000 gpm is modeled at the pump as an assigned flow pump.

The total pump head is shown in the AFT Fathom Pump Summary at the top of the Output window.

[List of All Verification Models](#)

Verification Case 18 Problem Statement

Verification Case 18

Igor Karassik, William Krutzsch, Warren Fraser, Joseph Messina, Pump Handbook, 2nd Ed., 1986, McGraw-Hill, Page 8.9-11, Example 1

Karassik, Krutzsch, Fraser and Messina Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

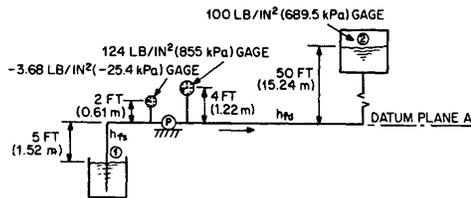


FIG. 8 Example 1.

expressed in feet (meters), the usual way of rating centrifugal pumps. For an explanation of positive displacement pump differential pressure, its use and relationship to pump total head, see Chap. 3.

Pump total head can be measured by installing gages at the pump suction and discharge connections and then substituting these gage readings into Eq. 4. Pump total head may also be found by measuring the energy difference between any two points in the pumping system, one on each side of the pump, providing all losses (other than pump losses) between these points are credited to the pump and added to the energy head difference. Therefore, between any two points in a pumping system where the energy is added only by the pump and the specific weight (force) of the liquid does not change (for example, as a result of temperature), the following general equation for determining pump total head applies:

$$TH = (H_2 - H_1) + \Sigma h_{f(1-2)} \quad (7)$$

$$= \left(\frac{V_2^2}{2g} + \frac{p_2}{\gamma} + Z_2 \right) - \left(\frac{V_1^2}{2g} + \frac{p_1}{\gamma} + Z_1 \right) + \Sigma h_{f(1-2)}$$

where the subscripts 1 and 2 denote points in the pumping system anywhere upstream and downstream from the pump, respectively, and

- H = total head of system, (+) or (-) ft (m) gage or (+) ft (m) abs
- V = velocity, ft/s (m/s)
- p = pressure, (+) or (-) lb/in² (N/m²) gage or (+) lb/in² (N/m²) abs
- Z = elevation above (+) or below (-) datum plane, ft (m)
- γ = specific weight (force) of liquid (assumed the same between points), lb/ft³ (N/m³)
- g = acceleration of gravity, 32.17 ft/s² (9.807 m/s²)
- Σh_f = sum of piping losses between points, ft (m)

When the specific gravity of the liquid is known, the pressure head may be calculated from the following relationships:

in feet
$$\frac{p}{\gamma} = \frac{0.016 \text{ lb/ft}^2}{\text{sp. gr.}} \text{ or } \frac{2.3 \text{ lb/in}^2}{\text{sp. gr.}} \quad (8a)$$

in meters
$$\frac{p}{\gamma} = \frac{0.102 \text{ kPa}}{\text{sp. gr.}} \text{ or } \frac{1.02 \times 10^{-3} \text{ bar}}{\text{sp. gr.}} \quad (8b)$$

The velocity in a pipe may be calculated as follows:

in feet per second
$$V = \frac{(\text{gm})(0.408)}{(\text{pipe ID in inches})^2} \quad (9a)$$

in meters per second
$$V = \frac{(\text{m}^3/\text{h})(3.54)}{(\text{pipe ID in cm})^2} \text{ or } \frac{(\text{liters/s})(12.7)}{(\text{pipe ID in cm})^2} \quad (9b)$$

The following example illustrates the use of Eqs. 4 and 7 for determining pump total head.

EXAMPLE 1 A centrifugal pump delivers 1000 gpm (227 m³/s) of liquid of specific gravity 0.8 from the suction tank to the discharge tank through the piping shown in Fig. 8. (a) Calculate

Verification Case 18 Problem Statement

8.10
PUMPING SYSTEMS

pump total head using gages and the datum plane selected. (b) Calculate total head using the pressures at points 1 and 2 and the same datum plane as (a).

Given: Suction pipe ID = 8 in (203 mm), discharge pipe ID = 6 in (152 mm), $h_f =$ pipe valve, and fitting losses, $h_{fs} = 3$ ft (0.91 m), $h_{fd} = 25$ ft (7.62 m).

in USCS units Calculated pipe velocity = $\frac{(\text{gpm})(0.408)}{(\text{ID in inches})^2}$

$$V_s = \frac{1000 \times 0.408}{8^2} = 6.38 \text{ ft/s}$$

$$V_d = \frac{1000 \times 0.408}{6^2} = 11.33 \text{ ft/s}$$

in SI units Calculated pipe velocity = $\frac{(\text{m}^3/\text{h})(3.54)}{(\text{ID in cm})^2}$

$$V_s = \frac{227 \times 3.54}{20.3^2} = 1.95 \text{ m/s}$$

$$V_d = \frac{227 \times 3.54}{15.2^2} = 3.48 \text{ m/s}$$

(a) From Eq. 4,

$$TH = \left(\frac{V_2^2}{2g} + \frac{p_2}{\gamma} + Z_2 \right) - \left(\frac{V_1^2}{2g} + \frac{p_1}{\gamma} + Z_1 \right)$$

and Eq. 8,

in USCS units $\frac{p}{\gamma} = \frac{2.31 \text{ lb/in}^2}{\text{sp. gr.}}$

in SI units $\frac{p}{\gamma} = \frac{0.102 \text{ kPa}}{\text{sp. gr.}}$

Therefore,

in USCS units

$$TH = \left(\frac{11.33^2}{2 \times 32.2} + \frac{2.31 \times 124}{0.8} + 4 \right) - \left(\frac{6.38^2}{2 \times 32.2} + \frac{2.31(-3.68)}{0.8} + 2 \right)$$

$$= 364 - (-8) = 372 \text{ ft} \cdot \text{lb/lb, or ft}$$

in SI units

$$TH = \left(\frac{3.48^2}{2 \times 9.807} + \frac{0.102 \times 855}{0.8} + 1.22 \right) - \left(\frac{1.95^2}{2 \times 9.807} + \frac{0.102(-25.4)}{0.8} + 2 \right)$$

$$= 110.9 - (-2.43) = 113.3 \text{ N} \cdot \text{m/N, or m}$$

(b) from Eq. 7,

$$TH = \left(\frac{V_2^2}{2g} + \frac{p_2}{\gamma} + Z_2 \right) - \left(\frac{V_1^2}{2g} + \frac{p_1}{\gamma} + Z_1 \right) + \Sigma h_{f(1-2)}$$

and Eq. 8,

in USCS units $\frac{p}{\gamma} = \frac{2.31 \text{ lb/in}^2}{\text{sp. gr.}}$

in SI units $\frac{p}{\gamma} = \frac{0.102 \text{ kPa}}{\text{sp. gr.}}$

Therefore

$$\begin{aligned} \text{in USCS units } TH &= \left(0 + \frac{2.31 \times 100}{0.8} + 50 \right) - (0 + 0 - 5) + (3 + 25) \\ &= 399 - (-5) + 28 = 372 \text{ ft}\cdot\text{lb/lb, or ft} \end{aligned}$$

$$\begin{aligned} \text{in SI units } TH &= \left(0 + \frac{0.102 \times 689.5}{0.8} + 15.24 \right) - (0 + 0 - 1.52) + (0.91 + 7.62) \\ &= 103.2 - (-1.52) + 8.53 = 113.3 \text{ N}\cdot\text{m/N, or m} \end{aligned}$$

ENERGY AND HYDRAULIC GRADIENTS

The total energy at any point in a pumping system may be calculated for a particular rate of flow using Bernoulli's equation (Eq. 1). If some convenient datum plane is selected and the total energy, or head, at various locations along the system is plotted to scale, the line drawn through these points is called the *energy gradient*. Figure 9 shows the variation in total energy H measured in feet (meters) from the suction liquid surface point 3 to the discharge liquid surface point 4. A horizontal energy gradient indicates no loss of head.

The line drawn through the sum of the pressure and elevation heads at various points represents the pressure variation in flow measured above the datum plane. It also represents the height the liquid would rise in vertical columns relative to the datum plane when the columns are placed at various locations along pipes having positive pressure anywhere in the system. This line, shown dotted in Fig. 9, is called the *hydraulic gradient*. The difference between the energy gradient line and the hydraulic gradient line is the velocity head in the pipe at that point.

The pump total head is the algebraic difference between the total energy at the pump discharge (point 2) and the total energy at the pump suction (point 1).

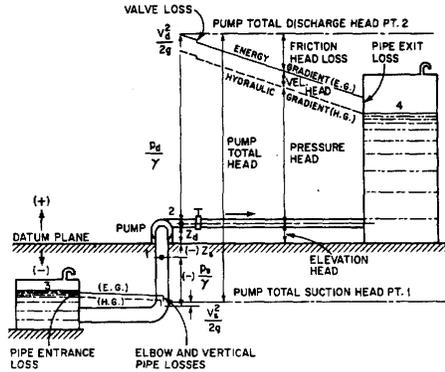


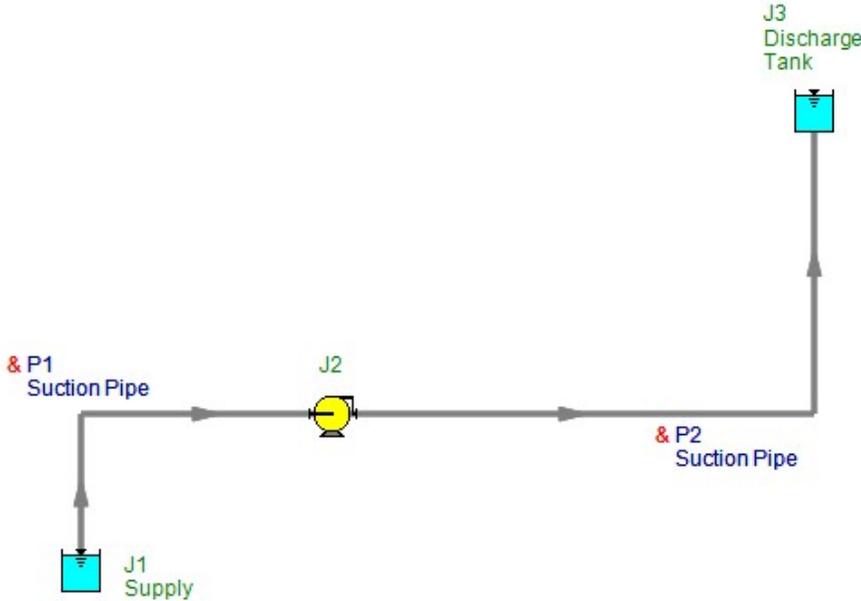
FIG. 9 Energy and hydraulic gradients.

SYSTEM-HEAD CURVES

A pumping system may consist of piping, valves, fittings, open channels, vessels, nozzles, weirs, meters, process equipment, and other liquid-handling conduits through which flow is required for various reasons. When a particular system is being analyzed for the purpose of selecting a pump or pumps, the resistance to flow of the liquid through these various components must be calculated. It will be explained in more detail later in this section that the resistance increases with flow at a rate approximately equal to the square of the flow through the system. In addition to over-

View Verification Case 18 Model

[Verification Case 18](#)



Verification Case 19

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify19.fth

REFERENCE: Igor Karassik, William Krutzsch, Warren Fraser, Joseph Messina, Pump Handbook, 2nd Ed., 1986, McGraw-Hill, Page 8.35-37, Example 6

FLUID: Water at 109 deg. F

ASSUMPTIONS: N/A

RESULTS:

Parameter	Handbook	AFT Fathom
Head loss (feet)	1.82	1.874

DISCUSSION:

The handbook uses a chart to obtain the friction factor of 0.012. AFT Fathom uses a more accurate correlation based on Colebrook-White to obtain a friction factor of 0.1235, which is 3% higher. This is the reason the AFT Fathom head loss calculation differs slightly from the handbook. The AFT Fathom calculation is more accurate because of the higher accuracy friction factor.

[List of All Verification Models](#)

Verification Case 19 Problem Statement

Verification Case 19

Igor Karassik, William Krutzsch, Warren Fraser, Joseph Messina, Pump Handbook, 2nd Ed., 1986, McGraw-Hill, Page 8.35-37, Example 6

Karassik, Krutzsch, Fraser and Messina Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

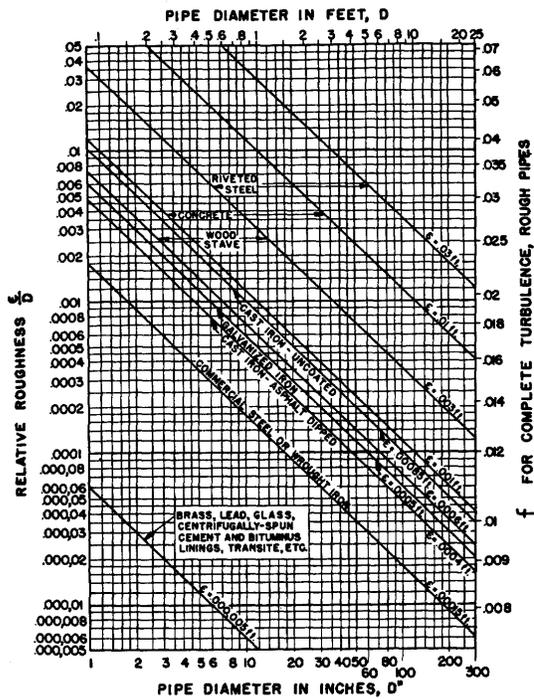


FIG. 32 Relative roughness and friction factors for new, clean pipes for flow of 60°F (15.6°C) water. (Ref. 5) (1 meter = 39.37 in = 3.28 ft)

EXAMPLE 6 Calculate the frictional head loss for 100 ft (30.48 m) of 20-in (50.8-cm), Schedule 40, 19.350-in (49.15-cm) ID, seamless steel pipe for 109°F (42.8°C) water flowing at a rate of 11,500 gpm (2612 m³/h). Use the Darcy-Weisbach formula.

$$\text{In USCS units } V = \frac{\text{gpm}}{(\text{pipe ID in inches})^2} \times 0.408 = \frac{11,500}{19.35^2} \times 0.408 = 12.53 \text{ ft/s}$$

$$VD^* = 12.53 \times 19.35 = 242 \text{ ft/s} \times \text{in}$$

$$\text{In SI units } V = \frac{\text{m}^3/\text{h}}{(\text{pipe ID in cm})^2} \times 3.54 = \frac{2612}{49.15^2} \times 3.54 = 3.83 \text{ m/s}$$

$$VD = 3.83 \times 0.4915 = 1.88 \text{ m/s} \times \text{m} = 1.88 \times 129.2 = 242 \text{ ft/s} \times \text{in}$$

From Fig. 33 $Re = 3 \times 10^6$

From Fig. 32 $\frac{\epsilon}{D} = 0.00009$

From Fig. 31 $f = 0.012$

Verification Case 19 Problem Statement

TABLE 2 Values of Friction Factor *C* to Be Used with the Hazen-Williams Formula in Fig. 34.

Type of pipe	Age	Size, in ^a	<i>C</i>
Cast iron	New	All sizes	130
		12 and over	120
	5 years old	8	119
		4	118
		24 and over	113
	10 years old	12	111
		4	107
		24 and over	100
	20 years old	12	96
		4	89
		30 and over	90
	30 years old	16	87
		4	75
		30 and over	83
40 years old	16	80	
	4	64	
	40 and over	77	
	24	74	
	4	55	
Welded steel	Any age, any size		Same as for cast iron pipe 5 years older
Riveted steel	Any age, any size		Same as for cast iron pipe 10 years older
Wood-stave	Average value, regardless of age and size		120
Concrete or concrete-lined	Large sizes, good workmanship, steel forms		140
	Large sizes, good workmanship, wooden forms		120
	Centrifugally spun		135
Vitrified	In good condition		110

^aIn × 25.4 = mm.
SOURCE: Adapted from Ref. 15.

Using Eq. 16,

in USCS units

$$D = \frac{19.35}{12} = 1.61 \text{ ft}$$

$$h_f = f \frac{L V^2}{d 2g} = 0.012 \frac{100}{1.61} \times \frac{12.53^2}{2 \times 32.17} = 1.82 \text{ ft}$$

in SI units

$$h_f = f \frac{L V^2}{D 2g} = 0.012 \frac{30.48}{0.4915} \times \frac{3.83^2}{2 \times 9.807} = 0.556 \text{ m}$$

EXAMPLE 7 The flow in Example 6 is increased until complete turbulence results. Determine the friction factor *f* and flow.

Verification Case 19 Problem Statement

8.38

PUMPING SYSTEMS

From Fig. 31, follow the relative roughness curve $\epsilon/D = 0.00009$ to the beginning of the zone marked "complete turbulence, rough pipes" and read

$$f = 0.0119 \quad \text{at } Re = 2 \times 10^7$$

The problem may also be solved using Fig. 32. Enter relative roughness $\epsilon/D = 0.00009$ and read directly across to

$$f = 0.0119$$

An increase in Re from 3×10^6 to 2×10^7 would require an increase in flow to

$$\text{in USCS units} \quad \frac{2 \times 10^7}{3 \times 10^6} \times 11,500 = 76,700 \text{ gpm}$$

$$\text{in SI units} \quad \frac{2 \times 10^7}{3 \times 10^6} \times 2612 = 17,413 \text{ m}^3/\text{h}$$

EXAMPLE 8 The liquid in Example 6 is changed to water at 60°F (15.6°C). Determine Re , f , and the frictional head loss per 100 ft (100 m) of pipe.

$$VD' = 242 \text{ ft/s} \times \text{in} \quad (\text{as in Example 6})$$

Since the liquid is 60°F (15.6°C) water, enter Fig. 31 and read directly downward from VD' to

$$Re = 1.8 \times 10^6$$

Where the line VD' to Re crosses $\epsilon/D = 0.00009$ in Fig. 31, read

$$f = 0.013$$

Water at 60°F (15.6°C) is more viscous than 109°F (42.8°C) water, and this accounts for the fact that Re decreases and f increases. Using Eq. 16, it can be calculated that the frictional head loss increases to

$$\text{in USCS units} \quad h_f = f \frac{L}{D} \frac{V^2}{2g} = 0.013 \frac{100}{1.61} \times \frac{12.53^2}{2 \times 32.17} = 1.97 \text{ ft}$$

$$\text{in SI units} \quad h_f = f \frac{L}{D} \frac{V^2}{2g} = 0.013 \frac{100}{0.4915} \times \frac{3.83^2}{2 \times 9.807} = 1.97 \text{ m}$$

EXAMPLE 9 A 102-in (259-cm) ID welded steel pipe is to be used to convey water at a velocity of 11.9 ft/s (3.63 m/s). Calculate the expected loss of head due to friction per 1000 ft and per 1000 m of pipe after 20 years. Use the empirical Hazen-Williams formula.

From Table 2, $C = 100$.

$$\begin{aligned} \text{in USCS units} \quad r &= D/4 = 102/(4 \times 12) = 2.13 \text{ ft} \\ \text{in SI units} \quad r &= D/4 = 2.59/4 = 0.648 \text{ m} \end{aligned}$$

Substituting in Eq. 17,

$$\text{in USCS units} \quad S^{0.54} = \frac{V}{1.318 C r^{0.63}} = \frac{11.9}{1.318 \times 100 \times 2.13^{0.63}} = 0.0557$$

$$S = (0.0557)^{1/0.54} = 0.0048 \text{ ft/ft}$$

$$h_f = 1000 \times 0.0048 = 4.8 \text{ ft}$$

$$\text{in SI units} \quad S^{0.54} = \frac{V}{0.8492 C r^{0.63}} = \frac{3.63}{0.8492 \times 100 \times 0.648^{0.63}} = 0.0562$$

$$S = (0.0562)^{1/0.54} = 0.0048 \text{ m/m}$$

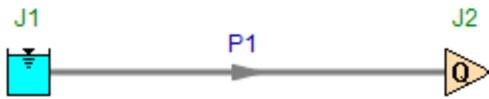
$$h_f = 1000 \times 0.0048 = 4.8 \text{ m}$$

The problem may also be solved by using Fig. 34, following the trace lines:

$$h_f \approx 5 \text{ ft (m)}$$

View Verification Case 19 Model

[Verification Case 19](#)



Verification Case 20

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify20.fth

REFERENCE: Igor Karassik, William Krutzsch, Warren Fraser, Joseph Messina, Pump Handbook, 2nd Ed., 1986, McGraw-Hill, Page 8.38, Example 8

FLUID: Water at 60 deg. F

ASSUMPTIONS: N/A

RESULTS:

Parameter	Handbook	AFT Fathom
Head loss (feet)	1.97	1.937

DISCUSSION:

The handbook uses a chart to obtain the Reynolds number of 1.8×10^6 and friction factor of 0.013. AFT Fathom solves for the Reynolds number and uses a more accurate correlation based on Colebrook-White to obtain a friction factor of 0.1277, which is 2% lower. This is the reason the AFT Fathom head loss calculation differs slightly from the handbook. The AFT Fathom calculation is more accurate because of the higher accuracy Reynolds number and friction factor.

[List of All Verification Models](#)

Verification Case 20 Problem Statement

Verification Case 20

Igor Karassik, William Krutzsch, Warren Fraser, Joseph Messina, Pump Handbook, 2nd Ed., 1986, McGraw-Hill, Page 8.38, Example 8

Karassik, Krutzsch, Fraser and Messina Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

8.38 PUMPING SYSTEMS

From Fig. 31, follow the relative roughness curve $\epsilon/D = 0.00009$ to the beginning of the zone marked "complete turbulence, rough pipes" and read

$$f = 0.0119 \quad \text{at } Re = 2 \times 10^7$$

The problem may also be solved using Fig. 32. Enter relative roughness $\epsilon/D = 0.00009$ and read directly across to

$$f = 0.0119$$

An increase in Re from 3×10^6 to 2×10^7 would require an increase in flow to

$$\text{in USCS units} \quad \frac{2 \times 10^7}{3 \times 10^6} \times 11,500 = 76,700 \text{ gpm}$$

$$\text{in SI units} \quad \frac{2 \times 10^7}{3 \times 10^6} \times 2612 = 17,413 \text{ m}^3/\text{h}$$

EXAMPLE 8 The liquid in Example 6 is changed to water at 60°F (15.6°C). Determine Re , f , and the frictional head loss per 100 ft (100 m) of pipe.

$$VD' = 242 \text{ ft/s} \times \text{in} \quad (\text{as in Example 6})$$

Since the liquid is 60°F (15.6°C) water, enter Fig. 31 and read directly downward from VD' to

$$Re = 1.8 \times 10^6$$

Where the line VD' to Re crosses $\epsilon/D = 0.00009$ in Fig. 31, read

$$f = 0.013$$

Water at 60°F (15.6°C) is more viscous than 109°F (42.8°C) water, and this accounts for the fact that Re decreases and f increases. Using Eq. 16, it can be calculated that the frictional head loss increases to

$$\text{in USCS units} \quad h_f = f \frac{L}{D} \frac{V^2}{2g} = 0.013 \frac{100}{1.61} \times \frac{12.53^2}{2 \times 32.17} = 1.97 \text{ ft}$$

$$\text{in SI units} \quad h_f = f \frac{L}{D} \frac{V^2}{2g} = 0.013 \frac{100}{0.4915} \times \frac{3.83^2}{2 \times 9.807} = 1.97 \text{ m}$$

EXAMPLE 9 A 102-in (259-cm) ID welded steel pipe is to be used to convey water at a velocity of 11.9 ft/s (3.63 m/s). Calculate the expected loss of head due to friction per 1000 ft and per 1000 m of pipe after 20 years. Use the empirical Hazen-Williams formula.

From Table 2, $C = 100$.

$$\begin{aligned} \text{in USCS units} \quad r &= D/4 = 102/(4 \times 12) = 2.13 \text{ ft} \\ \text{in SI units} \quad r &= D/4 = 2.59/4 = 0.648 \text{ m} \end{aligned}$$

Substituting in Eq. 17,

$$\text{in USCS units} \quad S^{0.54} = \frac{V}{1.318Cr^{0.63}} = \frac{11.9}{1.318 \times 100 \times 2.13^{0.63}} = 0.0557$$

$$S = (0.0557)^{1/0.54} = 0.0048 \text{ ft/ft}$$

$$h_f = 1000 \times 0.0048 = 4.8 \text{ ft}$$

$$\text{in SI units} \quad S^{0.54} = \frac{V}{0.8492Cr^{0.63}} = \frac{3.63}{0.8492 \times 100 \times 0.648^{0.63}} = 0.0562$$

$$S = (0.0562)^{1/0.54} = 0.0048 \text{ m/m}$$

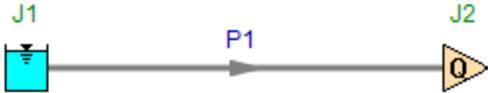
$$h_f = 1000 \times 0.0048 = 4.8 \text{ m}$$

The problem may also be solved by using Fig. 34, following the trace lines:

$$h_f \approx 5 \text{ ft (m)}$$

View Verification Case 20 Model

[Verification Case 20](#)



Verification Case 21

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify21.fth

REFERENCE: Igor Karassik, William Krutzsch, Warren Fraser, Joseph Messina, Pump Handbook, 2nd Ed., 1986, McGraw-Hill, Page 8.38, Example 9

FLUID: Water, using Hazen-Williams

ASSUMPTIONS: N/A

RESULTS:

Parameter	Handbook	AFT Fathom
Head loss (feet)	4.8	4.8

DISCUSSION:

Results for AFT Fathom vary somewhat from previous versions of AFT Fathom (prior to version 7) because the equation used to convert the Hazen-Williams factor to the Darcy-Weisbach friction factor was modified to use the traditional formula, as given in the AFT Fathom help file.

[List of All Verification Models](#)

Verification Case 21 Problem Statement

Verification Case 21

Igor Karassik, William Krutzsch, Warren Fraser, Joseph Messina, Pump Handbook, 2nd Ed., 1986, McGraw-Hill, Page 8.38, Example 9

Karassik, Krutzsch, Fraser and Messina Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

8.38 PUMPING SYSTEMS

From Fig. 31, follow the relative roughness curve $\epsilon/D = 0.00009$ to the beginning of the zone marked "complete turbulence, rough pipes" and read

$$f = 0.0119 \quad \text{at } Re = 2 \times 10^7$$

The problem may also be solved using Fig. 32. Enter relative roughness $\epsilon/D = 0.00009$ and read directly across to

$$f = 0.0119$$

An increase in Re from 3×10^6 to 2×10^7 would require an increase in flow to

$$\text{in USCS units} \quad \frac{2 \times 10^7}{3 \times 10^6} \times 11,500 = 76,700 \text{ gpm}$$

$$\text{in SI units} \quad \frac{2 \times 10^7}{3 \times 10^6} \times 2612 = 17,413 \text{ m}^3/\text{h}$$

EXAMPLE 8 The liquid in Example 6 is changed to water at 60°F (15.6°C). Determine Re , f , and the frictional head loss per 100 ft (100 m) of pipe.

$$VD' = 242 \text{ ft/s} \times \text{in} \quad (\text{as in Example 6})$$

Since the liquid is 60°F (15.6°C) water, enter Fig. 31 and read directly downward from VD' to

$$Re = 1.8 \times 10^6$$

Where the line VD' to Re crosses $\epsilon/D = 0.00009$ in Fig. 31, read

$$f = 0.013$$

Water at 60°F (15.6°C) is more viscous than 109°F (42.8°C) water, and this accounts for the fact that Re decreases and f increases. Using Eq. 16, it can be calculated that the frictional head loss increases to

$$\text{in USCS units} \quad h_f = f \frac{L}{D} \frac{V^2}{2g} = 0.013 \frac{100}{1.61} \times \frac{12.53^2}{2 \times 32.17} = 1.97 \text{ ft}$$

$$\text{in SI units} \quad h_f = f \frac{L}{D} \frac{V^2}{2g} = 0.013 \frac{100}{0.4915} \times \frac{3.83^2}{2 \times 9.807} = 1.97 \text{ m}$$

EXAMPLE 9 A 102-in (259-cm) ID welded steel pipe is to be used to convey water at a velocity of 11.9 ft/s (3.63 m/s). Calculate the expected loss of head due to friction per 1000 ft and per 1000 m of pipe after 20 years. Use the empirical Hazen-Williams formula.

From Table 2, $C = 100$.

$$\begin{aligned} \text{in USCS units} \quad r &= D/4 = 102/(4 \times 12) = 2.13 \text{ ft} \\ \text{in SI units} \quad r &= D/4 = 2.59/4 = 0.648 \text{ m} \end{aligned}$$

Substituting in Eq. 17,

$$\text{in USCS units} \quad S^{0.54} = \frac{V}{1.318Cr^{0.63}} = \frac{11.9}{1.318 \times 100 \times 2.13^{0.63}} = 0.0557$$

$$S = (0.0557)^{1/0.54} = 0.0048 \text{ ft/ft}$$

$$h_f = 1000 \times 0.0048 = 4.8 \text{ ft}$$

$$\text{in SI units} \quad S^{0.54} = \frac{V}{0.8492Cr^{0.63}} = \frac{3.63}{0.8492 \times 100 \times 0.648^{0.63}} = 0.0562$$

$$S = (0.0562)^{1/0.54} = 0.0048 \text{ m/m}$$

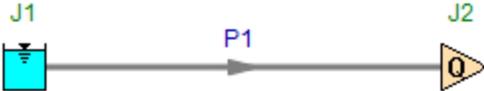
$$h_f = 1000 \times 0.0048 = 4.8 \text{ m}$$

The problem may also be solved by using Fig. 34, following the trace lines:

$$h_f \approx 5 \text{ ft (m)}$$

View Verification Case 21 Model

[Verification Case 21](#)



Verification Case 22

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify22.fth

REFERENCE: Igor Karassik, William Krutzsch, Warren Fraser, Joseph Messina, Pump Handbook, 2nd Ed., 1986, McGraw-Hill, Page 8.61-62, Example 13

FLUID: Oil

ASSUMPTIONS: N/A

RESULTS:

Parameter	Handbook	AFT Fathom
Head loss (feet)	98.34	98.58

DISCUSSION:

The viscosity was not provided, so the friction factors determined in the handbook were used directly for pipes 1 and 2. In addition, a loss factor for the 1 ½ inch gate valve was not in the AFT Fathom library, so was entered in pipe 2 as a separate loss factor on the Fittings & Losses tab.

The head loss is given in the Junction Deltas table at the top of the Output window. The junction delta was setup in the Output Control window. Or it can be calculated by adding up the pipe head losses.

[List of All Verification Models](#)

Verification Case 22 Problem Statement

[Verification Case 22](#)

Igor Karassik, William Krutzsch, Warren Fraser, Joseph Messina, Pump Handbook, 2nd Ed., 1986, McGraw-Hill, Page 8.61-62, Example 13

[Karassik, Krutzsch, Fraser and Messina Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

8.1 PUMPING SYSTEMS AND SYSTEM-HEAD CURVES

8.61

$$\begin{aligned} \text{in SI units} \quad \Sigma h_f &= h_{fs} + h_{fd} + h_{f1} + h_{f2} + h_{f3} + h_{f4} + h_{f5} \\ &= 0.56 + 28.39 + 0.0078 + 0.044 + 0.0250 + 0.0803 \\ &\quad + 1.06 = 30.17 \text{ m} \\ \text{Total variation} &= \pm(0.044 + 0.0062 + 0.02 + 0.32) = \pm 0.39 \text{ m} \end{aligned}$$

EXAMPLE 13 Solve Example 12 using resistance coefficients from Tables 6.

Suction pipe:

$$\begin{aligned} \text{in USCS units} \quad h_{fs} &= 1.84 \text{ ft} \quad (\text{same as in Example 12}) \\ \text{in SI units} \quad h_{fs} &= 0.56 \text{ m} \quad (\text{same as in Example 12}) \end{aligned}$$

Discharge pipe:

$$\begin{aligned} \text{in USCS units} \quad h_{fd} &= 92.9 \text{ ft} \quad (\text{same as in Example 12}) \\ \text{in SI units} \quad h_{fd} &= 28.39 \text{ m} \quad (\text{same as in Example 12}) \end{aligned}$$

Valve and fitting losses from Tables 6 and Eq. 20: 2-in (51-mm) bellmouth, $K = 0.04$

$$\begin{aligned} \text{in USCS units} \quad h_{f1} &= 0.04 \frac{5.73^2}{2 \times 32.17} = 0.020 \text{ ft} \\ \text{in SI units} \quad h_{f1} &= 0.04 \frac{1.75^2}{2 \times 9.807} = 0.0062 \text{ m} \end{aligned}$$

2-in (51-mm) SR 90° elbow, $K = 30 f_T$

$$\begin{aligned} f_T &= 0.019 \quad (\text{from Table 5a}) \\ K &= 30 \times 0.019 = 0.57 \end{aligned}$$

$$\begin{aligned} \text{in USCS units} \quad h_{f2} &= 0.57 \frac{5.73^2}{2 \times 32.17} = 0.29 \text{ ft} \\ \text{in SI units} \quad h_{f2} &= 0.57 \frac{1.75^2}{2 \times 9.807} = 0.089 \text{ m} \end{aligned}$$

2-in (51-mm) gate valve, $\beta = 1$, $\theta = 0$, $K = 8 f_T$

$$K = 8 \times 0.019 = 0.15$$

$$\begin{aligned} \text{in USCS units} \quad h_{f3} &= 0.15 \frac{5.73^2}{2 \times 32.17} = 0.077 \text{ ft} \\ \text{in SI units} \quad h_{f3} &= 0.15 \frac{1.75^2}{2 \times 9.807} = 0.023 \text{ m} \end{aligned}$$

1½-in (38-mm) gate valve, $\beta = 1$, $\theta = 0$, $K = 8 f_T$ for Schedule 80

$$\begin{aligned} K &= 8 f_T (1.10/1.500)^4 = 10.62 f_T \text{ for Schedule 40} \\ f_T &= 0.021 \quad (\text{from Table 6a}) \\ K &= 10.62 \times 0.021 = 0.22 \end{aligned}$$

$$\begin{aligned} \text{in USCS units} \quad h_{f4} &= 0.22 \frac{9.44^2}{2 \times 32.17} = 0.30 \text{ ft} \\ \text{in SI units} \quad h_{f4} &= 0.22 \frac{2.88^2}{2 \times 9.807} = 0.093 \text{ m} \end{aligned}$$

Verification Case 22 Problem Statement

8.62

PUMPING SYSTEMS

1½-in (38-mm) swing check valve, $K = 100f_T$ (from Table 6b)

Minimum pipe velocity for full disk lift = $35\sqrt{V} = 35\sqrt{0.0189} = 4.81 \text{ ft/s} < 9.44 \text{ ft/s}$

$$K = 100 \times 0.021 = 2.1$$

in USCS units
$$h_{fs} = 2.1 \frac{9.44^2}{2 \times 32.17} = 2.91 \text{ ft}$$

in SI units
$$h_{fs} = 2.1 \frac{2.88^2}{2 \times 9.807} = 0.89 \text{ m}$$

Total pipe, valve, and fitting losses:

in USCS units
$$\begin{aligned} \Sigma h_f &= h_{fs} + h_{fd} + h_{f1} + h_{f2} + h_{fs} + h_{f4} + h_{fs} \\ &= 1.84 + 92.9 + 0.020 + 0.29 + 0.077 + 0.30 + 2.91 \\ &= 98.34 \text{ ft} \end{aligned}$$

in SI units
$$\begin{aligned} \Sigma h_f &= h_{fs} + h_{fd} + h_{f1} + h_{f2} + h_{fs} + h_{f4} + h_{fs} \\ &= 0.56 + 28.39 + 0.062 + 0.089 + 0.023 + 0.093 + 0.89 \\ &= 30.05 \text{ m} \end{aligned}$$

INCREASERS The head lost when there is a sudden increase in pipe diameter, with velocity changing from V_1 to V_2 in the direction of flow, can be calculated analytically. Computed results have been confirmed experimentally to be true to within $\pm 3\%$. The head loss is expressed as shown below, with K computed to be equal to unity:

$$h = K \frac{(V_1 - V_2)^2}{2g} = K \left[1 - \left(\frac{D_1}{D_2} \right)^2 \right]^2 \frac{V_1^2}{2g} = K \left[\left(\frac{D_2}{D_1} \right)^2 - 1 \right]^2 \frac{V_2^2}{2g} \quad (23)$$

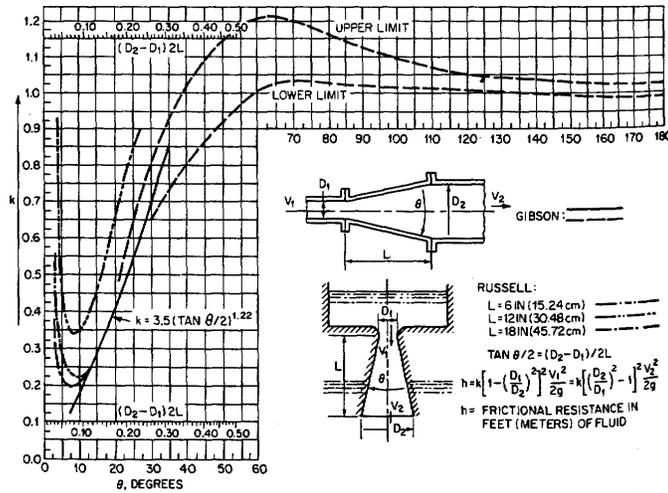
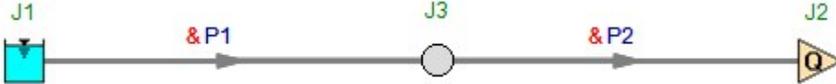


FIG. 39 Resistance coefficients for increasers and difusers. ($D =$ in or ft [m]; $V =$ ft/s [m/s]). (Ref. 5)

View Verification Case 22 Model

[Verification Case 22](#)



Verification Case 23

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify23.fth

REFERENCE: Hydraulic Institute, Effects of Liquid Viscosity on Rotodynamic (Centrifugal and Vertical) Pump Performance, ANSI/HI 9.6.7-2010, 2010, Hydraulic Institute, Page 13

FLUID: Viscous fluid through a pump

ASSUMPTIONS: N/A

RESULTS:

Parameter	HI Standard	AFT Fathom
CQ	0.938	0.9381
CH	0.938	0.9380
CE	0.739	0.7388
CH at 60% Flow	0.958	0.9577

DISCUSSION:

The kinematic viscosity in centistokes (120 cSt) was converted to dynamic viscosity for use in AFT Fathom assuming a specific gravity of 0.9. This results in a viscosity of 108 cp. Discharge and suction pressures and pipes were used, with the pipe diameters made very large to minimize pressure loss.

The water flow rate and pressure difference from the problem statement were used to find an approximate CQ of 0.9395, which provides a viscous flow rate of 413.38 gal/min for the 100 percent flow case and 248.028 gal/min for the 60 percent flow case. These were used in assigned flow junctions which replaced the inlet assigned pressures for the final calculation. The pumps were redefined with the water pump curve from the problem statement, shown below, to find the final values of CQ, CH, and CE.

Water Flow Rate (gal/min)	Water Head (ft)	Water Efficiency (Decimal)
264	340	0.602
352	323	0.660
440	300	0.680
528	272	0.660

The correction factors, CQ, CH and CE, all correspond closely to those in the handbook. The value of CE given in the reference problem was shown as 0.738 in the calculation, but 0.739 in Table 9.6.7.4.5b. In addition, CE is calculated as 0.739 if calculated using the same numbers as given in Step 4, so it is assumed that the difference in the value given in the reference includes round-off errors.

The CQ, CH, and CE values in AFT Fathom are displayed in the Pump Summary and within the labels on the Workspace.

[List of All Verification Models](#)

Verification Case 23 Problem Statement

[Verification Case 23](#)

Hydraulic Institute, Effects of Liquid Viscosity on Rotodynamic (Centrifugal and Vertical) Pump Performance, ANSI/HI 9.6.7-2010, 2010, Hydraulic Institute, Page 13

[Hydraulic Institute ANSI HI 9.6.7-2010 Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

HI Effects of Liquid Viscosity on Pump Performance — 2010

EXAMPLE (US customary units): Refer to Figure 9.6.7.4.5d, Example performance chart of a single-stage pump, and Table 9.6.7.4.5b, Example calculations. The given single-stage pump has a water performance best efficiency flow of 440 gpm at 300 ft total head at 3550 rpm and has a pump efficiency of 0.68. The procedure below illustrates how to correct the pump performance characteristics for a viscous liquid of 120 cSt and specific gravity of 0.90.

Step 1. Calculate parameter B based on the water performance best efficiency flow conditions using Equation 3. If the pump is a multistage configuration, calculate parameter B using the head per stage.

Given units of Q_{BEP-W} in gpm, H_W in ft, N in rpm, and V_{vis} in cSt:

$$B = 26.6 \times \frac{(120)^{0.50} \times (300)^{0.0625}}{(440)^{0.375} \times (3550)^{0.25}} = 5.50$$

Step 2. Calculate correction factor for flow (C_Q) using Equation 4 and correct the flows corresponding to ratios of water best efficiency flow (Q_W / Q_{BEP-W}).

$$C_Q = (2.71)^{-0.165 \times (\log 5.50)^{3.16}} = 0.938$$

$$\text{At: } \frac{Q_W}{Q_{BEP-W}} = 1.00$$

$$Q_{vis} = 0.938 \times 440 \times 1.00 = 413 \text{ gpm}$$

$$\text{At: } \frac{Q_W}{Q_{BEP-W}} = 0.60$$

$$Q_{vis} = 0.938 \times 440 \times 0.60 = 248 \text{ gpm}$$

From Equation 5, the correction factor for head (C_{BEP-H}) is equal to (C_Q) at Q_{BEP-W}

$$Q_{BEP-H} = C_Q = 0.938$$

At Q_{BEP-W} , the corresponding viscous head ($H_{BEP-vis}$) is:

$$H_{BEP-vis} = 0.938 \times 300 = 281 \text{ ft}$$

Step 3. Calculate head correction factors (C_H) and corresponding values of viscous head (H_{vis}) for flows (Q_W) greater than or less than the water best efficiency flow (Q_{BEP-W}).

At 60% of Q_{BEP-W} the corresponding head correction factor (C_H) and viscous head (H_{vis}) are calculated using Equation 6:

$$C_H = 1 - (1 - 0.938) \times (0.60)^{0.75} = 0.958$$

$$H_{vis} = 0.958 \times 340 = 326 \text{ ft}$$

Step 4. Calculate the correction factor for efficiency (C_η) and the corresponding values of viscous pump efficiency (η_{vis}) for flows (Q_W) greater than, less than, and equal to the water best efficiency flow Q_{BEP-W} . Equation 7 is used to calculate C_η as the value of parameter $B = 5.50$ calculated in Step 1 falls within the range of 1 to 40:

$$C_\eta = (5.50)^{-[0.0547 \times (5.50)^{0.69}]} = 0.738$$

Verification Case 23 Problem Statement

HI Effects of Liquid Viscosity on Pump Performance — 2010

$$\text{At: } \frac{Q_W}{Q_{BEP-W}} = 1.00$$

Where: $\eta_W = 0.680$

$$\eta_{vis} = 0.738 \times 0.680 = 0.502$$

$$\text{At: } \frac{Q_W}{Q_{BEP-W}} = 0.60$$

Where: $\eta_W = 0.602$

$$\eta_{vis} = 0.738 \times 0.602 = 0.444$$

Step 5. Calculate the values for viscous pump shaft input power (P_{vis}) for flows (Q_W) greater than, less than, or equal to the water best efficiency flow Q_{BEP-W} using Equation 9:

$$\text{At: } \frac{Q_W}{Q_{BEP-W}} = 1.00$$

$$P_{vis} = \frac{413 \times 281 \times 0.90}{3960 \times 0.502} = 52.5 \text{ hp}$$

$$\text{At: } \frac{Q_W}{Q_{BEP-W}} = 0.60$$

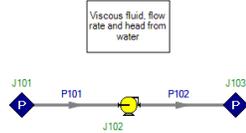
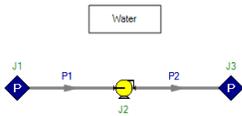
$$P_{vis} = \frac{248 \times 326 \times 0.90}{3960 \times 0.444} = 41.4 \text{ hp}$$

Table 9.6.7.4.5b — Example calculations (US customary units)

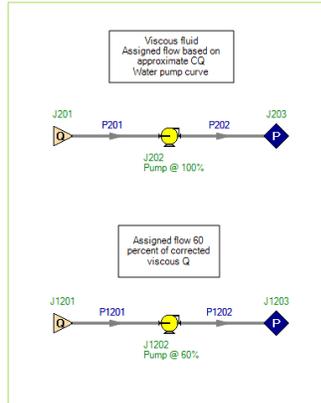
Viscosity of liquid to be pumped (V_{vis}) — cSt	120			
Specific gravity of viscous liquid (s)	0.90			
Pump shaft speed (N) — rpm	3550			
Ratio of water best efficiency flow Q_W / Q_{BEP-W}	0.60	0.80	1.00	1.20
Water rate of flow (Q_W or Q_{BEP-W}) — gpm	264	352	440	528
Water head per stage (H_W) — ft	340	323	300	272
Water pump efficiency (η_W)	0.602	0.66	0.680	0.66
Parameter B	5.50			
Correction factor for flow (C_Q)	0.938			
Correction factors for head (C_H or C_{BEP-H})	0.958	0.948	0.938	0.929
Correction factor for efficiency (C_η)	0.739			
Corrected flow (Q_{vis}) — gpm	248	330	413	495
Corrected head per stage (H_{vis} or $H_{BEP-vis}$) — ft	326	306	281	252
Corrected efficiency (η_{vis})	0.44	0.49	0.50	0.49
Viscous shaft input power (P_{vis}) — bhp	41.4	46.7	52.5	57.9

View Verification Case 23 Model

Verification Case 23



This provides an approximate C_d of 0.3395 and thus a viscous flowrate of 413.38 gpm



Verification Case 24

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify24.fth

REFERENCE: Michael R. Lindeburg, P.E., Mechanical Engineering Review Manual, 7th Ed., Professional Publications, Page 3-19, 21, Example 3.19

FLUID: Water

ASSUMPTIONS: N/A

RESULTS:

Parameter	Lindenbug	AFT Fathom
Head loss (feet)	51.6	51.53

DISCUSSION:

All input was provided explicitly by the problem statement.

Lindeburg pulls the friction factor directly from the Moody chart, while AFT Fathom uses the Colebrook-White equation to iterate for friction factor. Both methods result in a friction factor of 0.0195 if the friction factor is rounded to 4 digits.

[List of All Verification Models](#)

Verification Case 24 Problem Statement

Verification Case 24

Michael R. Lindeburg, P.E., Mechanical Engineering Review Manual, 7th Ed., Professional Publications, Page 3-19, 21, Example 3.19

Lindeburg Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

PART 4: Fluid Dynamics

1 FLUID CONSERVATION LAWS

Many fluid flow problems can be solved by using the principles of conservation of mass and energy.

When applied to fluid flow, the principle of mass conservation is known as the *continuity equation*:

$$\rho_1 A_1 v_1 = \rho_2 A_2 v_2 \quad 3.66$$

$$\dot{m}_1 = \dot{m}_2 \quad 3.67$$

If the fluid is incompressible, $\rho_1 = \rho_2$, so

$$A_1 v_1 = A_2 v_2 \quad 3.68$$

$$\dot{V}_1 = \dot{V}_2 \quad 3.69$$

The energy conservation principle is based on the Bernoulli equation. However, terms for friction loss and hydraulic machines must be included.

$$\left(\frac{p_1}{\rho} + \frac{v_1^2}{2g_c} + z_1 \right) + h_A = \left(\frac{p_2}{\rho} + \frac{v_2^2}{2g_c} + z_2 \right) + h_E + h_f \quad 3.70$$

2 HEAD LOSS DUE TO FRICTION

The most common expression for calculating head loss due to friction (h_f) is the *Darcy formula*:

$$h_f = \frac{fLv^2}{2Dg_c} \quad 3.71$$

The *Moody friction factor chart* (figure 3.13) probably is the most convenient method of determining the friction factor, f .

The basic parameter required to use the Moody friction factor chart is the Reynolds number. If the Reynolds number is less than 2000, the friction factor is given by equation 3.72.

$$f = \frac{64}{N_{Re}} \quad 3.72$$

For turbulent flow ($N_{Re} > 2000$), the friction factor depends on the relative roughness of the pipe. This roughness is expressed by the ratio $\frac{\epsilon}{D}$, where ϵ is the specific surface roughness and D is the inside diameter. Values of ϵ for various types of pipe are found in table 3.8.

Another method for finding the friction head loss is the *Hazen-Williams formula*. The Hazen-Williams formula gives good results for liquids that have kinematic viscosities around $1.2 \text{ EE}-5 \text{ ft}^2/\text{sec}$ (corresponding to 60°F water). At extremely high and low temperatures, the Hazen-Williams formula can be as much as 20% in er-

ror for water. The Hazen-Williams formula should be used only for turbulent flow.

The Hazen-Williams head loss is

$$h_f = \frac{(3.022)(v)^{1.85} L}{(C)^{1.85} (D)^{1.165}} \quad 3.73$$

Or, in terms of other units,

$$h_f = (10.44)(L) \frac{(gpm)^{1.85}}{(C)^{1.85} (d_{inches})^{4.8665}} \quad 3.74$$

Use of these formulas requires a knowledge of the Hazen-Williams coefficient, C , which is assumed to be independent of the Reynolds number. Table 3.8 gives values of C for various types of pipe.

Values of f and h_f are appropriate for clean, new pipe. As some pipes age, it is not uncommon for scale build-up to decrease the equivalent flow diameter. This diameter decrease produces a dramatic increase in the friction loss.

$$\frac{h_{f, \text{scaled}}}{h_{f, \text{new}}} = \left(\frac{D_{\text{new}}}{D_{\text{scaled}}} \right)^5 \quad 3.75$$

Because of this scale effect, an uprating factor of 10-30% is commonly applied to f or h_f in anticipation of future service conditions.

Example 3.19

50°F water is pumped through 4" schedule 40 welded steel pipe ($\epsilon = .0002$) at the rate of 300 gpm. What is the friction head loss calculated by the Darcy formula for 1000 feet of pipe?

First, it is necessary to collect data on the pipe and water. The fluid viscosity and pipe dimensions can be found from tables at the end of the chapter.

$$\begin{aligned} \text{kinematic viscosity} &= 1.41 \text{ EE} - 5 \text{ ft}^2/\text{sec} \\ \text{inside diameter} &= .3355 \text{ ft} \\ \text{flow area} &= .0884 \text{ ft}^2 \end{aligned}$$

The flow quantity is

$$(300)(.002228) = .6684 \text{ cfs}$$

The velocity is

$$v = \frac{\dot{V}}{A} = \frac{.6684}{.0884} = 7.56 \text{ fps}$$

The Reynolds number is

$$N_{Re} = \frac{(3355)(7.56)}{1.41 \text{ EE} - 5} = 1.8 \text{ EE} 5$$

The relative roughness is

$$\frac{\epsilon}{D} = \frac{.0002}{.3355} = .0006$$

From the Moody friction factor chart, $f = .0195$.

From equation 3.71,

$$h_f = \frac{(.0195)(1000)(7.56)^2}{(2)(.3355)(32.2)} = 51.6 \text{ ft}$$

Example 3.20

Repeat example 3.19 using the Hazen-Williams formula. Assume $C = 100$.

Using equation 3.73,

$$h_f = \frac{(3.012)(7.56)^{1.85}(1000)}{(100)^{1.85}(.3355)^{4.8655}} = 90.5 \text{ ft}$$

Using equation 3.74,

$$h_f = (10.44)(1000) \frac{(300)^{1.85}}{(100)^{1.85}(4.026)^{4.8655}} = 90.9 \text{ ft}$$

3 MINOR LOSSES

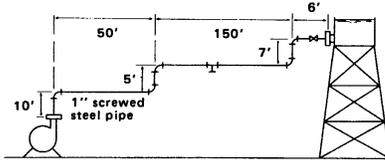
In addition to the head loss caused by friction between the fluid and the pipe wall, losses also are caused by obstructions in the line, changes in direction, and changes in flow area. These losses are named *minor losses* because they are much smaller in magnitude than the h_f term. Two methods are used to determine these losses: the method of equivalent lengths and the method of loss coefficients.

The method of *equivalent lengths* uses a table to convert each valve and fitting into an equivalent length of straight pipe. This length is added to the actual pipeline length and substituted into the Darcy equation for L_e .

$$h_f = \frac{f L_e v^2}{2 D g_c} \quad 3.76$$

Example 3.21

Using table 3.9, determine the equivalent length of the piping network shown.



The line consists of:

1 gate valve	.84
5 90° standard elbows	5.2 X 5
1 tee run	3.2
straight pipe	<u>228</u>
$L_e =$	258 feet

The alternative is to use a loss coefficient, K . This loss coefficient, when multiplied by the velocity head, will give the head loss in feet. This method must be used to find exit and entrance losses.

$$h_f = K \frac{v^2}{2 g_c} \quad 3.77$$

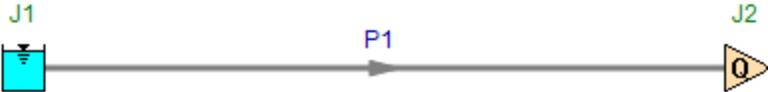
Table 3.9
Typical Equivalent Lengths of Schedule 40 Straight Pipe
For Steel Fittings and Valves
 (For any fluid in turbulent flow)

Fitting Type	Pipe Size*				
	1"	2"	4"	6" (flanged pipe)	8"
Short Radius 90° Elbow	5.2	8.5	13.0	8.9	12.0
Long Radius 90° Elbow	2.7	3.6	4.6	5.7	7.0
Regular 45° Elbow	1.3	2.7	5.5	5.6	7.7
Tee, flow through line (run)	3.2	7.7	17.0	3.8	4.7
Tee, flow through stem	6.6	12.0	21.0	18.0	24.0
180° Return Bend	5.2	8.5	13.0	8.9	12.0
Globe Valve	29.0	54.0	110.0	190.0	260.0
Gate Valve	.84	1.5	2.5	3.2	3.2
Angle Valve	17.0	18.0	18.0	63.0	90.0
Swing Check Valve	11.0	19.0	38.0	63.0	90.0
Coupling or Union	.29	.45	.65	—	—

*Screwed pipe and fittings unless flanged indicated.

View Verification Case 24 Model

[Verification Case 24](#)



Verification Case 25

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify25.fth

REFERENCE: Michael R. Lindeburg, P.E., Mechanical Engineering Review Manual, 7th Ed., Professional Publications, Page 3-21, Example 3.20

FLUID: Water

ASSUMPTIONS: N/A

RESULTS:

Parameter	Lindenbug	AFT Fathom
Head loss (feet)	90.5	90.66

DISCUSSION:

Results for AFT Fathom vary somewhat from previous versions of AFT Fathom (prior to version 7) because the equation used to convert the Hazen-Williams factor to the Darcy-Weisbach friction factor was modified to use the traditional formula, as given in the AFT Fathom help file.

[List of All Verification Models](#)

Verification Case 25 Problem Statement

Verification Case 25

Michael R. Lindeburg, P.E., Mechanical Engineering Review Manual, 7th Ed., Professional Publications, Page 3-21, Example 3.20

Lindeburg Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

The relative roughness is

$$\frac{\epsilon}{D} = \frac{.0002}{.3355} = .0006$$

From the Moody friction factor chart, $f = .0195$.

From equation 3.71,

$$h_f = \frac{(.0195)(1000)(7.56)^2}{(2)(.3355)(32.2)} = 51.6 \text{ ft}$$

Example 3.20

Repeat example 3.19 using the Hazen-Williams formula. Assume $C = 100$.

Using equation 3.73,

$$h_f = \frac{(3.012)(7.56)^{1.85}(1000)}{(100)^{1.85}(.3355)^{1.165}} = 90.5 \text{ ft}$$

Using equation 3.74,

$$h_f = (10.44)(1000) \frac{(300)^{1.85}}{(100)^{1.85}(4.026)^{4.8655}} = 90.9 \text{ ft}$$

3 MINOR LOSSES

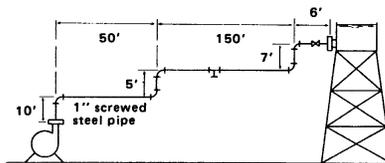
In addition to the head loss caused by friction between the fluid and the pipe wall, losses also are caused by obstructions in the line, changes in direction, and changes in flow area. These losses are named *minor losses* because they are much smaller in magnitude than the h_f term. Two methods are used to determine these losses: the method of equivalent lengths and the method of loss coefficients.

The method of *equivalent lengths* uses a table to convert each valve and fitting into an equivalent length of straight pipe. This length is added to the actual pipeline length and substituted into the Darcy equation for L_e .

$$h_f = \frac{fL_e v^2}{2Dg_c} \quad 3.76$$

Example 3.21

Using table 3.9, determine the equivalent length of the piping network shown.



The line consists of:

1 gate valve	.84
5 90° standard elbows	5.2 X 5
1 tee run	3.2
straight pipe	<u>228</u>
$L_e =$	<u>258 feet</u>

The alternative is to use a loss coefficient, K . This loss coefficient, when multiplied by the velocity head, will give the head loss in feet. This method must be used to find exit and entrance losses.

$$h_f = K \frac{v^2}{2g_c} \quad 3.77$$

Table 3.9
Typical Equivalent Lengths of Schedule 40 Straight Pipe
For Steel Fittings and Valves
(For any fluid in turbulent flow)

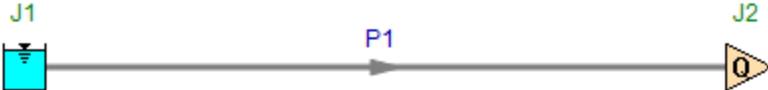
Fitting Type	Equivalent Length, ft				
	1"	Pipe Size* 2"	4"	6" (flanged pipe)	8"
Short Radius 90° Elbow	5.2	8.5	13.0	8.9	12.0
Long Radius 90° Elbow	2.7	3.6	4.6	5.7	7.0
Regular 45° Elbow	1.3	2.7	5.5	5.6	7.7
Tee, flow through line (run)	3.2	7.7	17.0	3.8	4.7
Tee, flow through stem	6.6	12.0	21.0	18.0	24.0
180° Return Bend	5.2	8.5	13.0	8.9	12.0
Globe Valve	29.0	54.0	110.0	190.0	260.0
Gate Valve	.84	1.5	2.5	3.2	3.2
Angle Valve	17.0	18.0	18.0	63.0	90.0
Swing Check Valve	11.0	19.0	38.0	63.0	90.0
Coupling or Union	.29	.45	.65	—	—

*Screwed pipe and fittings unless flanged indicated.

PROFESSIONAL ENGINEERING REGISTRATION PROGRAM • P.O. Box 911, San Carlos, CA 94070

View Verification Case 25 Model

[Verification Case 25](#)



Verification Case 26

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify26.fth

REFERENCE: Michael R. Lindenbug, P.E., Mechanical Engineering Review Manual, 7th Ed., Professional Publications, Page 4-6, Example 4.1

FLUID: Water

ASSUMPTIONS: N/A

RESULTS:

Parameter	Lindenbug	AFT Fathom
NPSHA (feet)	50.7	50.69
Total suction head (feet)	Not given	18.38

DISCUSSION:

The NPSHA is given in the AFT Fathom Pump Summary of the Output window.

[List of All Verification Models](#)

Verification Case 26 Problem Statement

Verification Case 26

Michael R. Lindeburg, P.E., Mechanical Engineering Review Manual, 7th Ed., Professional Publications, Page 4-6, Example 4.1

Lindeburg Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

4-6

HYDRAULIC MACHINES

step 2: Calculate NPSHA from either equation 4.12 or 4.13.

step 3: If NPSHA is greater than NPSHR, cavitation will not occur. A good safety margin is 2–3 feet of fluid. If NPSHA is insufficient, it should be increased or the NPSHR should be decreased. NPSHA can be increased by:

- Increasing the height of the free fluid level of the supply tank
- reducing the distance and minor losses between the supply tank and the pump, or by using a larger pipe size
- reducing the temperature of the fluid
- pressurizing the supply tank
- reducing the flow rate or velocity to reduce friction in the suction line

NPSHR can be reduced by:

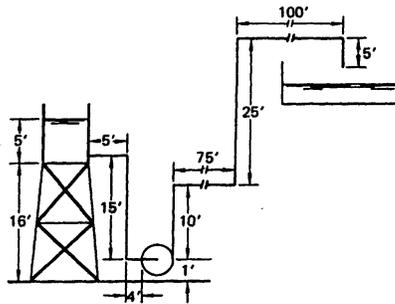
- placing a throttling valve in the discharge line (i.e., this will increase the total head, thereby reducing the capacity of the pump and driving its operating point into a region of lower NPSHR)
- using a double suction pump

Applications which require very high NPSHR, such as boiler feed pumps needing 150 to 250 feet, should use booster pumps in front of the high NPSHR pumps. Such booster pumps are typically single stage, double suction pumps running at low speed. Their NPSHR can be 25 feet or less.

It is important to note that throttling the input line to a pump and venting or evacuating the receiving tank both work to increase cavitation. Throttling the input line increases the friction head term and decreases NPSHA. Evacuating the receiving tank increases the flow rate, which also increases the friction head term.

Example 4.1

2 cfs of water are pumped from a feed tank mounted on a platform to an open reservoir through 6" schedule 40 ($c/D = .000293$) steel pipe. Determine the static suction head, total suction head, and NPSHA.



step 1: Assume 60°F and 14.7 psia. From equation 4.4,

$$h_a = \frac{(14.7)(144)}{62.4} = 33.9 \text{ ft}$$

step 2: For 6" schedule 40 steel pipe, $D = .505$ ft, $A = .201$ ft².

step 3:

$$v = \frac{Q}{A} = \frac{2}{.201} = 9.95 \text{ ft/sec}$$

step 4: The equivalent lengths of the pipe and flanged fittings are:

square entrance loss	$1 \times 16 = 16$
90° long radius elbows	$2 \times 5.7 = 11.4$
pipe run (5 + 15 + 4)	<u>24</u>
	51.4 ft

step 5: At 60°, the kinematic viscosity of water is 1.217 EE-5 ft²/sec. The vapor pressure is .6 feet of water.

step 6: The Reynolds number is

$$N_{Re} = \frac{(.505)(9.95)}{1.217 \text{ EE} - 5} = 4.13 \text{ EE}5$$

step 7: From the Moody friction factor chart, $f = .0165$, so

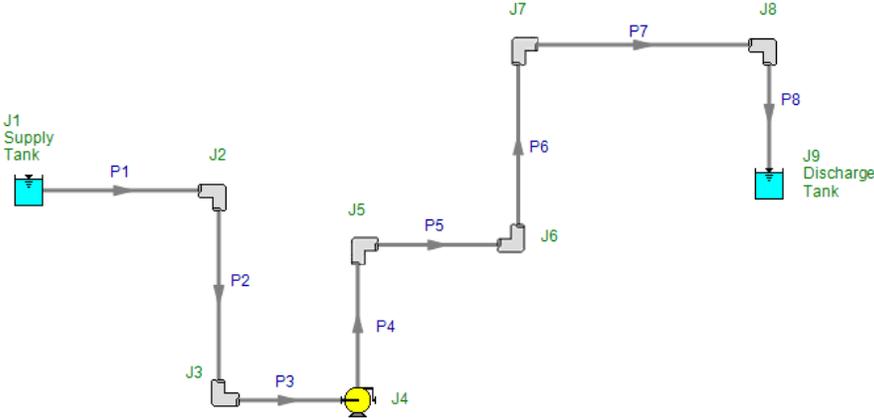
$$h_f = \frac{(.0165)(51.4)(9.95)^2}{(2)(.505)(32.2)} = 2.6 \text{ ft}$$

step 8: From equation 4.12,

$$\text{NPSHA} = 33.9 + 20 - 2.6 - .6 = 50.7 \text{ ft}$$

View Verification Case 26 Model

[Verification Case 26](#)



Verification Case 27

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify27.fth

REFERENCE: Michael R. Lindeburg, P.E., Mechanical Engineering Review Manual, 7th Ed., Professional Publications, Page 5-6, Example 5.6

FLUID: Air

ASSUMPTIONS: Air is at 70 deg. F, inlet pressure is 5 inches of water, gauge, wall roughness is 0.00538 inches

RESULTS:

Parameter	Lindenbug	AFT Fathom
Velocity (feet/min)	2200	2170
Pressure loss(in. water per 100 feet)	0.5	0.4867

DISCUSSION:

The reference solves the problem using a chart. No roughness data for the duct is given. Using the chart on page 5-7 of the reference, a representative roughness was determined as follows: for a 12 inch duct at 3000 feet/min, the pressure drop is 1 inch of water per 100 feet. This information was entered into the AFT Pipe Sizing Utility (PSU, part of the Engineering Utility Suite) to obtain an effective duct roughness. A value of 0.00538 inches was obtained, and this was used in AFT Fathom. Results from PSU are given below.

Slight differences in predictions above result from the reference's round-off errors and reading the solution from a chart.

- PSU Results
 - Friction Factor = 0.01788
 - Epsilon = 0.00538 inches
 - Hazen-Williams Factor = 110.390
- Input Parameters
 - Delta Pressure = 1.000 in. H₂O std.
 - Velocity = 3000 feet/min
 - Vol. Flow Rate = 39.270 ft³/sec
 - Mass Flow Rate = 2.941 lbm/sec
 - Reynolds Number = 306548
- Geometry: Cylindrical Pipe
 - Diameter = 12 inches
 - Area = 0.78540 ft²

- Wetted Perimeter = 37.699 inches
- Length = 100 feet
- Fluid = Air @ 1 atm (vapor)
 - Temperature = 70 deg. F
 - Density = 0.0749 lbm/ft³
 - Dynamic Viscosity = 0.04398 lbm/hr-ft

[List of All Verification Models](#)

Verification Case 27 Problem Statement

Verification Case 27

Michael R. Lindeburg, P.E., Mechanical Engineering Review Manual, 7th Ed., Professional Publications, Page 5-6, Example 5.6

Lindeburg Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

5-6

FANS AND DUCTWORK

PART 2: Duct Design

1 FRICTION LOSSES IN DUCTS

Friction losses can be calculated from the Moody equation.

$$f_{\text{loss}} = \frac{fLp_v}{D} \approx (0.0270) \frac{L}{(d)^{1.22}} \left(\frac{v}{1000} \right)^{1.82} \quad 5.27$$

$$\approx (3.9 \text{ EE} - 9)(v)^{2.43} \frac{L}{(Q)^{.81}} \quad 5.28$$

In practice, equation 5.27 seldom is used. Figure 5.4 is based on equation 5.27 with a value of specific roughness equal to .0005, a standard density of .075 lbm/ft³, clean round galvanized metal ductwork, and approximately 40 joints per 100 feet. The chart can be used for temperatures between 50 °F and 90 °F. For operation outside this range, the pressure loss should be corrected with equation 5.29. K_v is usually taken as 1.0.

$$(P_{\text{loss}})_{\text{actual}} = P_{\text{loss, Eq. 5.4}} \times \frac{K_v}{K_d} \quad 5.29$$

Figure 5.4 is for use with standard round, galvanized ducts. Multiply the friction losses by the factors in table 5.2 for other materials. (Actual values are velocity dependent. Tables and charts exist for this purpose.)

Table 5.2

Multiplicative Factors for Non-Standard Ducts

Smooth ducts—no joints	.6-.95
Smooth concrete	1.1-1.4
Rough concrete/good brick	1.2-1.8

The *equivalent diameter* of a rectangular air duct with dimensions a and b , and aspect ratio less than 8.0 is

$$D_e = 1.3 \frac{(ab)^{.825}}{(a+b)^{.25}} \quad 5.30$$

A round duct with diameter D_e will have the same friction and capacity as a square duct with dimensions a and b . Figure 5.4 can be used with D_e and flow rate to find the friction loss.

If the aspect ratio of the rectangular duct is known, a round duct can be converted to a rectangular duct of equal friction. The *aspect ratio*, which should be kept below 8.0 for ease of manufacturing, is

$$R = \frac{\text{longest side}}{\text{short side}} \quad 5.31$$

The short side, a , is given by equation 5.32.

$$a = \frac{D_e(R+1)^{\frac{1}{2}}}{1.3(R)^{.825}} \quad 5.32$$

Example 5.6

2000 cfm of air flow in a 13" diameter duct. What is the velocity and the friction loss per 100 feet of duct?

Although the $Q = Av$ relationship could be used to find the velocity, it is expedient to use figure 5.4. By locating the intersection of the 2000 cfm and the 13" lines, the velocity is found to be 2200 fpm.

Dropping straight down from the intersection point to the horizontal scale gives the friction loss as approximately .5" w.g. per 100 feet.

Example 5.7

What size duct is required to carry 2000 cfm at 1600 fpm?

Figure 5.4 shows that a 15" diameter duct is required.⁴ The friction loss is approximately 0.23" w.g. per 100 feet.

2 MINOR AND DYNAMIC LOSSES

Minor losses are fairly independent of air velocity and roughness. In the *loss coefficient method*, the losses are calculated as a percentage of the velocity pressure.

$$p = c \left(\frac{v}{4005} \right)^2 = cp_v \quad 5.33$$

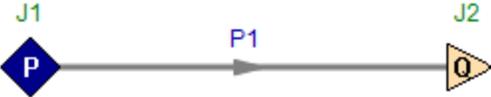
Typical values of c are given in table 5.3. Subscripts 1 and 2 refer to upstream and downstream, respectively. The coefficient c always should be used with the velocity at the point corresponding to its subscript.

The *equivalent length method* also can be used to calculate the friction of a bend or an elbow. As with equivalent lengths used in liquid flow problems, each obstruction produces a frictional loss equivalent to some length of duct. These lengths are given in multiples of the duct diameter in table 5.3.

⁴Any size duct can be manufactured. However, there are standard sizes available, and these sizes should be chosen to minimize cost. Generally, every whole-inch size up to 30" diameter is available, although some odd-number sizes may be premium-priced. After 30", sizes are available in 2" increments.

View Verification Case 27 Model

[Verification Case 27](#)



Verification Case 28

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify28.fth

REFERENCE: Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-1, Example 4-1

FLUID: Water at 80 deg. F

ASSUMPTIONS: Supply pressure is from a 10 feet tank

RESULTS:

Parameter	Crane	AFT Fathom
Reynolds number	89,600	88,780
Friction factor	0.0182	0.01845

[List of All Verification Models](#)

Verification Case 28 Problem Statement

[Verification Case 28](#)

Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-1, Example 4-1

[Crane Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Examples of Flow Problems

Theory and answers to questions regarding proper application of formulas to flow problems can be presented to good advantage by the solution of practical problems. A few simple flow problems were presented in Chapter 3 to illustrate the use of the nomographs. Other problems, both simple and complex, are presented in this chapter.

Many of the examples given in this chapter employ the basic formulas of Chapters 1 and 2; these formulas were rewritten in more commonly used terms for Chapter 3. Use of nomographs, when applicable, are indicated in the solution of these problems.

The controversial subject regarding the selection of a formula most applicable to the flow of gas through long pipe lines is analyzed in Chapter 1. It is shown that the three commonly used formulas are basically identical, the only difference being in the selection of friction factors. A comparison of results obtained, using the three formulas, is presented in this chapter.

An original method has been developed for the solution of problems involving the discharge of compressible fluids from pipe systems. Illustrative examples applying this method demonstrate the simplicity of handling these, heretofore complex, problems.

CHAPTER 4

4-1

Reynolds Number and Friction Factor For Pipe Other Than Steel

The example below shows the procedure in obtaining the Reynolds number and friction factor for smooth pipe (plastic). The same procedure applies for any pipe other than steel such as concrete, wood stave, riveted steel, etc. For relative roughness of these and other piping materials, see page A-23.

Example 4-1 . . . Smooth Pipe (Plastic)

Given: Water at 80 F is flowing through 70 feet of 2-inch standard wall plastic pipe (smooth wall) at a rate of 50 gallons per minute.

Find: The Reynolds number and friction factor.

Solution:

1. $R_e = \frac{50.6 Q \rho}{d \mu}$ page 3-2

2. $\rho = 62.220$ page A-6

3. $d = 2.067$ page B-16

4. $\mu = 0.85$ page A-3

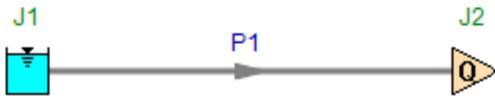
5. $R_e = \frac{50.6 \times 50 \times 62.220}{2.067 \times 0.85}$

$R_e = 89\,600$ or 8.96×10^4

6. $f = 0.0182$ for smooth pipe page A-24

View Verification Case 28 Model

[Verification Case 28](#)



Verification Case 29

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify29.fth

REFERENCE: Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-3, Example 4-6

FLUID: Water at 60 deg. F

ASSUMPTIONS: Friction factor in pipe is 0.018.

RESULTS:

Parameter	Crane	AFT Fathom
Velocity (feet/sec)	8.5	8.545
Flow Rate (gal/min)	196	196.9

DISCUSSION:

The loss factor for the flanged ball valve was not in the AFT Fathom library, and the resulting K factor of 0.58 was entered directly as a fitting and loss value for pipe 1.

Slight differences in the predictions result from round-off.

[List of All Verification Models](#)

Verification Case 29 Problem Statement

Verification Case 29

Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-3, Example 4-6

Crane Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Check Valves Determination of Size

Example 4-5... Lift Check Valves

Given: A globe type lift check valve with a wing-guided disc is required in a 3-inch Schedule 40 horizontal pipe carrying 70 F water at the rate of 80 gallons per minute.

Find: The proper size check valve and the pressure drop. The valve should be sized so that the disc is fully lifted at the specified flow; see page 2-7 for discussion.

Solution:

- $$v_{\min} = 40\sqrt{V} \dots \text{page A-27}$$

$$v = \frac{0.408 Q}{d^2} \dots \text{page 3-2}$$

$$\Delta P = \frac{18 \times 10^{-6} K \rho Q^2}{d^4} \dots \text{page 3-4}$$

$$K_1 = 600 f_T \dots \text{page A-27}$$

$$K_2 = \frac{K_1 + \beta [0.5(1 - \beta^2) + (1 - \beta^2)^2]}{\beta^4} \dots \text{page A-27}$$

$$\beta = \frac{d_1}{d_2} \dots \text{page A-26}$$
- $$\beta = 2.469 \dots \text{for } 2\frac{1}{2}'' \text{ Sched. 40 pipe; page B-16}$$

$$d_2 = 3.068 \dots \text{for } 3'' \text{ Sched. 40 pipe; page B-16}$$

$$V = 0.01605 \dots \text{70 F water; page A-6}$$

$$\rho = 62.305 \dots \text{70 F water; page A-6}$$

$$f_T = 0.018 \dots \text{for } 2\frac{1}{2}'' \text{ or } 3'' \text{ size; page A-26}$$
- $$v_{\min} = 40\sqrt{0.01605} = 5.1$$

$$v = \frac{0.408 \times 80}{3.068^2} = 3.5 \dots \text{for } 3'' \text{ valve}$$

Inasmuch as v is less than v_{\min} , a 3-inch valve will be too large. Try a $2\frac{1}{2}$ -inch size.

$$v = \frac{0.408 \times 80}{2.469^2} = 5.35 \dots \text{for } 2\frac{1}{2}'' \text{ valve}$$

Based on above, a $2\frac{1}{2}$ -inch valve installed in 3-inch Schedule 40 pipe with reducers is advisable.
- $$\beta = \frac{2.469}{3.068} = 0.80$$

$$\beta^2 = 0.64$$

$$\beta^4 = 0.41$$
- $$K_2 = \frac{600 \times 0.018 + 8 [0.5(1 - 0.64) + (1 - 0.64)^2]}{0.41}$$

$$K_2 = 27$$
- $$\Delta P = \frac{18 \times 10^{-6} \times 27 \times 62.305 \times 80^2}{3.068^4} = 2.2$$

Reduced Port Valves Velocity and Rate of Discharge

Example 4-6... Reduced Port Ball Valve

Given: Water at 60 F discharges from a tank with 22-feet average head to atmosphere through:

- 200 feet—3" Schedule 40 pipe
- 6—3" standard 90° threaded elbows
- 1—3" flanged ball valve having a $2\frac{3}{8}$ " diameter seat, 16° conical inlet, and 30° conical outlet end. Sharp-edged entrance is flush with the inside of the tank.

Find: Velocity of flow in the pipe and rate of discharge in gallons per minute.

Solution:

- $$h_L = K \frac{v^2}{2g} \text{ or } v = \sqrt{\frac{2gh_L}{K}} \dots \text{page 3-4}$$

$$v = 0.408 \frac{Q}{d^2} \text{ or } Q = 2.451 v d^2 \dots \text{page 3-2}$$
- $$K = 0.5 \dots \text{entrance; page A-29}$$

$$K = 1.0 \dots \text{exit; page A-29}$$

$$f_T = 0.018 \dots \text{page A-26}$$
- For K (ball valve), page A-28 indicates use of Formula 5. However, when inlet and outlet angles (θ) differ, Formula 5 must be expanded to:

$$K_2 = \frac{K_1 + 8 \sin^2 \frac{\theta}{2} (1 - \beta^2) + 2.6 \sin^2 \frac{\theta}{2} (1 - \beta^2)^2}{\beta^4}$$
- $$\beta = \frac{d_1}{d_2} = \frac{2.375}{3.068} = 0.77 \dots \text{page A-26}$$
- $$\sin \theta/2 = \sin 8^\circ = 0.14 \dots \text{valve inlet}$$
- $$\sin \theta/2 = \sin 15^\circ = 0.26 \dots \text{valve outlet}$$
- $$K_2 = \frac{3 \times 0.018 + 0.8 \times 0.14 (1 - 0.77^2) + 2.6 \times 0.26 (1 - 0.77^2)^2}{0.77^4} + 0.58 \dots \text{valve}$$

$$K = 6 \times 30 f_T = 180 \times 0.018 = 3.24 \dots \text{6 elbows; p. A-29}$$

$$K = f \frac{L}{D} = \frac{0.018 \times 200 \times 12}{3.068} = 14.08 \text{ pipe; p. 3-4}$$
- Then, for entire system (entrance, pipe, ball valve, six elbows, and exit),

$$K = 0.5 + 14.08 + 0.58 + 3.24 + 1.0 = 19.4$$
- $$v = \sqrt{(64.4 \times 22) + 19.4} = 8.5$$

$$Q = 2.451 \times 8.5 \times 3.068^2 = 196$$
- Calculate Reynolds number to verify that friction factor of 0.018 (zone of complete turbulence) is correct for flow condition... or, use "vd" scale at top of Friction Factor chart on page A-25.

$$vd = 8.5 \times 3.068 = 26$$
- Enter chart on page A-25 at $vd = 26$. Note f for 3-inch pipe is less than 0.02. Therefore, flow is in the transition zone (slightly less than fully turbulent) but the difference is small enough to forego any correction of K for the pipe.

View Verification Case 29 Model

[Verification Case 29](#)



Verification Case 30

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify30.fth

REFERENCE: Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-4, Example 4-7

FLUID: SAE 10 Lube Oil at 60F

ASSUMPTIONS: N/A

RESULTS:

Parameter	Crane	AFT Fathom
Volumetric flow rate (gal/min)	118	121.4
Velocity (feet/sec)	5.13	5.267
Reynolds number	1040	1095

DISCUSSION:

Because of the high oil viscosity, there is laminar flow in the pipe. As discussed in Crane, this is an iterative problem. Crane performs one iteration, while AFT Fathom performs numerous iterations. The difference in results is Crane's single iteration.

[List of All Verification Models](#)

Verification Case 30 Problem Statement

Verification Case 30

Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-4, Example 4-7

Crane Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Laminar Flow in Valves, Fittings, and Pipe

In flow problems where viscosity is high, calculate the Reynolds Number to determine whether the flow is laminar or turbulent.

Example 4-7

Given: S.A.E. 10 Lube Oil at 60 F flows through the system described in Example 4-6 at the same differential head.

Find: The velocity in the pipe and rate of flow in gallons per minute.

Solution:

$$1. \quad h_L = K \frac{v^2}{2g} \quad \dots \text{page 3-4}$$

$$v = \sqrt{\frac{2gh_L}{K}}$$

$$v = 0.408 \frac{Q}{d^2} \quad \dots \text{page 3-2}$$

$$Q = 2.451 \, v d^2$$

$$R_e = 124 \frac{dv\rho}{\mu} \quad \dots \text{page 3-2}$$

$$f = \frac{64}{R_e} \quad \dots \text{pipe, laminar flow; page 3-2}$$

$$K = f \frac{L}{D} \quad \dots \text{pipe; page 3-4}$$

$$2. \quad K_2 = 0.58 \quad \dots \text{valve; Example 4-6}$$

$$K = 3.24 \quad \dots \text{6 elbows; Example 4-6}$$

$$K = 0.5 \quad \dots \text{entrance; Example 4-6}$$

$$K = 1.0 \quad \dots \text{exit; Example 4-6}$$

$$\rho = 54.64 \quad \dots \text{page A-7}$$

$$\mu = 100 \quad \dots \text{page A-3}$$

$$h_L = 22 \quad \dots \text{Example 4-6}$$

$$3. \quad \text{*Assume laminar flow with } v = 5.$$

$$R_e = \frac{124 \times 3.068 \times 5 \times 54.64}{100} = 1040$$

$$f = 64 + 1040 = 0.062 \quad \dots \text{pipe}$$

$$K = \frac{0.062 \times 200 \times 12}{3.068} = 48.5 \quad \dots \text{pipe}$$

$$K = 48.5 + 0.58 + 3.24 + 0.5 + 1.0$$

$$K = 53.8 \quad \dots \text{entire system}$$

$$4. \quad v = \sqrt{\frac{64.4 \times 22}{53.8}} = 5.13$$

$$5. \quad Q = 2.451 \times 5.13 \times 3.068^2 = 118$$

*Note: This problem has two unknowns and, therefore, requires a trial-and-error solution. Two or three trial assumptions will usually bring the solution and final assumption into agreement within desired limits.

Example 4-8

Given: S.A.E. 70 Lube Oil at 100 F is flowing at the rate of 600 barrels per hour through 200 feet of 8-inch Schedule 40 pipe, in which an 8-inch conventional globe valve with full area seat is installed.

Find: The pressure drop due to flow through the pipe and valve.

Solution:

$$1. \quad \Delta P = \frac{8.82 \times 10^{-6} K \rho B^2}{d^4} \quad \dots \text{page 3-4}$$

$$R_e = \frac{35.4 \rho B}{d \mu} \quad \dots \text{page 3-2}$$

$$K_1 = 340 f_T \quad \dots \text{valve; page A-27}$$

$$K = f \frac{L}{D} \quad \dots \text{pipe; page 3-4}$$

$$f = \frac{64}{R_e} \quad \dots \text{pipe}$$

$$2. \quad S = 0.916 \text{ at } 60 \text{ F} \quad \dots \text{page A-7}$$

$$S = 0.90 \text{ at } 100 \text{ F} \quad \dots \text{page A-7}$$

$$d = 7.981 \quad \dots \text{8" Sched. 40 pipe; page B-17}$$

$$\mu = 470 \quad \dots \text{page A-3}$$

$$f_T = 0.014 \quad \dots \text{page A-26}$$

$$3. \quad \rho = 62.371 \times 0.90 = 56.1$$

$$R_e = \frac{35.4 \times 600 \times 56.1}{7.981 \times 470} = 318$$

$$R_e < 2000; \text{ therefore flow is laminar.}$$

$$4. \quad f = \frac{64}{318} = 0.20 \quad \dots \text{pipe}$$

$$K_1 = 340 \times 0.014 = 4.76 \quad \dots \text{valve}$$

$$K = \frac{0.20 \times 200 \times 12}{7.981} = 60.14 \quad \dots \text{pipe}$$

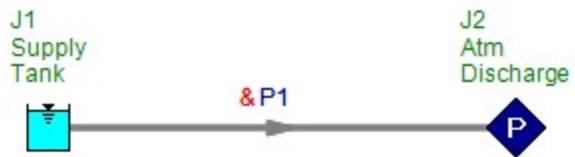
$$K = 4.76 + 60.14 = 64.9 \quad \dots \text{total system}$$

$$5. \quad \Delta P = \frac{8.82 \times 10^{-6} \times 64.9 \times 56.1 \times 600^2}{7.981^4}$$

$$\Delta P = 2.85$$

View Verification Case 30 Model

[Verification Case 30](#)



Verification Case 31

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: [FthVerify31.fth](#)FthVerify31.fth

REFERENCE: Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-4, Example 4-8

FLUID: SAE 70 Lube Oil at 100 deg. F

ASSUMPTIONS: N/A

RESULTS:

Parameter	Crane	AFT Fathom
Pressure drop (psid)	2.85	2.866

DISCUSSION:

Because of the high oil viscosity, there is laminar flow in the pipe.

[List of All Verification Models](#)

Verification Case 31 Problem Statement

Verification Case 31

Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-4, Example 4-8

Crane Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Laminar Flow in Valves, Fittings, and Pipe

In flow problems where viscosity is high, calculate the Reynolds Number to determine whether the flow is laminar or turbulent.

Example 4-7

Given: S.A.E. 10 Lube Oil at 60 F flows through the system described in Example 4-6 at the same differential head.

Find: The velocity in the pipe and rate of flow in gallons per minute.

Solution:

$$1. \quad h_L = K \frac{v^2}{2g} \quad \dots \text{page 3-4}$$

$$v = \sqrt{\frac{2gh_L}{K}}$$

$$v = 0.408 \frac{Q}{d^2} \quad \dots \text{page 3-2}$$

$$Q = 2.451 \text{ vd}^2$$

$$R_e = 124 \frac{dv\rho}{\mu} \quad \dots \text{page 3-2}$$

$$f = \frac{64}{R_e} \quad \dots \text{pipe, laminar flow; page 3-2}$$

$$K = f \frac{L}{D} \quad \dots \text{pipe; page 3-4}$$

$$2. \quad K_2 = 0.58 \quad \dots \text{valve; Example 4-6}$$

$$K = 3.24 \quad \dots \text{6 elbows; Example 4-6}$$

$$K = 0.5 \quad \dots \text{entrance; Example 4-6}$$

$$K = 1.0 \quad \dots \text{exit; Example 4-6}$$

$$\rho = 54.64 \quad \dots \text{page A-7}$$

$$\mu = 100 \quad \dots \text{page A-3}$$

$$h_L = 22 \quad \dots \text{Example 4-6}$$

$$3. \quad \text{*Assume laminar flow with } v = 5.$$

$$R_e = \frac{124 \times 3.068 \times 5 \times 54.64}{100} = 1040$$

$$f = 64 + 1040 = 0.062 \quad \dots \text{pipe}$$

$$K = \frac{0.062 \times 200 \times 12}{3.068} = 48.5 \quad \dots \text{pipe}$$

$$K = 48.5 + 0.58 + 3.24 + 0.5 + 1.0$$

$$K = 53.8 \quad \dots \text{entire system}$$

$$4. \quad v = \sqrt{\frac{64.4 \times 22}{53.8}} = 5.13$$

$$5. \quad Q = 2.451 \times 5.13 \times 3.068^2 = 118$$

*Note: This problem has two unknowns and, therefore, requires a trial-and-error solution. Two or three trial assumptions will usually bring the solution and final assumption into agreement within desired limits.

Example 4-8

Given: S.A.E. 70 Lube Oil at 100 F is flowing at the rate of 600 barrels per hour through 200 feet of 8-inch Schedule 40 pipe, in which an 8-inch conventional globe valve with full area seat is installed.

Find: The pressure drop due to flow through the pipe and valve.

Solution:

$$1. \quad \Delta P = \frac{8.82 \times 10^{-6} K \rho B^2}{d^4} \quad \dots \text{page 3-4}$$

$$R_e = \frac{35.4 \rho B}{d \mu} \quad \dots \text{page 3-2}$$

$$K_1 = 340 f_T \quad \dots \text{valve; page A-27}$$

$$K = f \frac{L}{D} \quad \dots \text{pipe; page 3-4}$$

$$f = \frac{64}{R_e} \quad \dots \text{pipe}$$

$$2. \quad S = 0.916 \text{ at } 60 \text{ F} \quad \dots \text{page A-7}$$

$$S = 0.90 \text{ at } 100 \text{ F} \quad \dots \text{page A-7}$$

$$d = 7.981 \quad \dots \text{8" Sched. 40 pipe; page B-17}$$

$$\mu = 470 \quad \dots \text{page A-3}$$

$$f_T = 0.014 \quad \dots \text{page A-26}$$

$$3. \quad \rho = 62.371 \times 0.90 = 56.1$$

$$R_e = \frac{35.4 \times 600 \times 56.1}{7.981 \times 470} = 318$$

$$R_e < 2000; \text{ therefore flow is laminar.}$$

$$4. \quad f = \frac{64}{318} = 0.20 \quad \dots \text{pipe}$$

$$K_1 = 340 \times 0.014 = 4.76 \quad \dots \text{valve}$$

$$K = \frac{0.20 \times 200 \times 12}{7.981} = 60.14 \quad \dots \text{pipe}$$

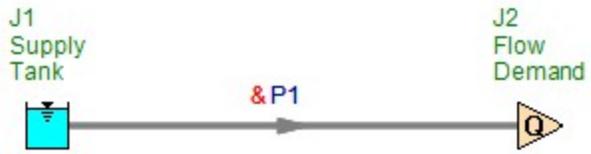
$$K = 4.76 + 60.14 = 64.9 \quad \dots \text{total system}$$

$$5. \quad \Delta P = \frac{8.82 \times 10^{-6} \times 64.9 \times 56.1 \times 600^2}{7.981^4}$$

$$\Delta P = 2.85$$

View Verification Case 31 Model

[Verification Case 31](#)



Verification Case 32

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify32.fth

REFERENCE: Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-5, Example 4-9

FLUID: SAE 70 Lube Oil at 100 deg. F

ASSUMPTIONS: Inlet pressure is 200 psig, and the gate valve is located 100 feet from the inlet (this does not affect the answer)

RESULTS:

Parameter	Crane	AFT Fathom
Pressure drop (psid)	56.6	56.67

DISCUSSION:

Because of the high oil viscosity, there is laminar flow in the pipes. The total pressure change in AFT Fathom can be found in two places. First, the discharge pressure (143.3 psig) can be subtracted from the inlet pressure (200 psig). Second, the pressure difference is given in the Junction Deltas table at the top of the Output window. The junction delta was setup in the Output Control window.

[List of All Verification Models](#)

Verification Case 32 Problem Statement

Verification Case 32

Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-5, Example 4-9

Crane Title Page

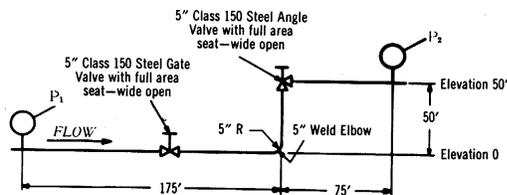
Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Laminar Flow in Valves, Fittings, and Pipe — continued

In flow problems where viscosity is high, calculate the Reynolds Number to determine whether the flow is laminar or turbulent.

Example 4-9

Given: S.A.E. 70 Lube Oil at 100 F is flowing through 5-inch Schedule 40 pipe at a rate of 600 gallons per minute, as shown in the following sketch.



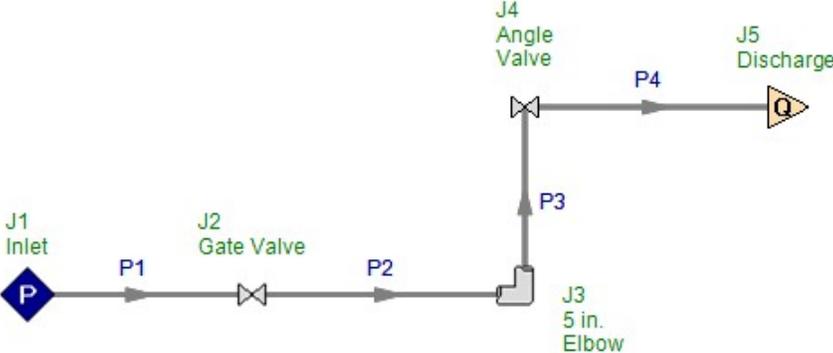
Find: The velocity in feet per second and pressure difference between gauges P_1 and P_2 .

Solution:

1. $v = \frac{0.408Q}{d^2}$ page 3-2
2. $R_e = \frac{50.6Q\rho}{d\mu}$ page 3-2
3. $\Delta P = \frac{18 \times 10^{-6} K \rho Q^2}{d^4}$ loss due to flow; page 3-4
4. $\Delta P = \frac{h_L \rho}{144}$ loss due to elevation change; page 3-5
5. $K_1 = 8 f_T$ gate valve; page A-27
6. $K_1 = 150 f_T$ angle valve; page A-27
7. $K = 20 f_T$ elbow; page A-29
8. $K = f \frac{L}{D}$ pipe; page 3-4
9. $f = \frac{64}{R_e}$ pipe; page 3-2
10. $d = 5.047$ 5" Sched. 40 pipe; page B-17
11. $S = 0.016$ at 60 F page A-7
12. $S = 0.90$ at 100 F page A-7
13. $\mu = 470$ page A-3
14. $\rho = 62.371 \times 0.90 = 56.1$
15. $f_T = 0.016$
16. $R_e = \frac{50.6 \times 600 \times 56.1}{5.047 \times 470} = 718$
17. $R_e < 2000$; therefore flow is laminar.
18. $f = \frac{64}{718} = 0.089$
19. Summarizing K for the entire system (gate valve, angle valve, elbow, and pipe),
 $K = (8 \times 0.016) + (150 \times 0.016) + (20 \times 0.016) + \frac{(0.089 \times 300 \times 12)}{5.047} = 66.3$
20. $v = \frac{0.408 \times 600}{5.047^2} = 9.6$
21. $\Delta P = \frac{18 \times 10^{-6} \times 66.3 \times 56.1 \times 600^2}{5.047^4} + \frac{50 \times 56.1}{144}$
22. $\Delta P = 56.6$ total

View Verification Case 32 Model

[Verification Case 32](#)



Verification Case 33

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: [FthVerify33.fth](#)FthVerify33.fth

REFERENCE: Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-6, Example 4-11

FLUID: Water at 180 deg. F

ASSUMPTIONS: Inlet pressure is 100 psig.

RESULTS:

Parameter	Crane	AFT Fathom
Pressure drop (psid)	1.91	1.91

DISCUSSION:

The total pressure change in AFT Fathom can be found in two places. First, the discharge pressure (98.09 psig) can be subtracted from the inlet pressure (100 psig). Second, the pressure difference is given in the Junction Deltas table at the top of the Output window. The junction delta was setup in the Output Control window.

[List of All Verification Models](#)

Verification Case 33 Problem Statement

Verification Case 33

Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-6, Example 4-11

Crane Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Pressure Drop and Velocity in Piping Systems

Example 4-10 . . . Piping Systems—Steam

Given: 600 psig steam at 850 F flows through 400 feet of horizontal 6-inch Schedule 80 pipe at a rate of 90,000 pounds per hour.

The system contains three 90 degree weld elbows having a relative radius of 1.5, one fully-open 6 x 4-inch Class 600 venturi gate valve as described in Example 4-4, and one 6-inch Class 600 y-pattern globe valve. Latter has a seat diameter equal to 0.9 of the inside diameter of Schedule 80 pipe, disc fully lifted.

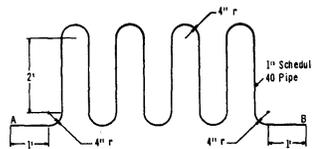
Find: The pressure drop through the system.

Solution:

- $\Delta P = \frac{28 \times 10^{-8} KW^2 \sqrt{V}}{d^5}$ page 3-4
- For globe valve (see page A-27),
 $K_2 = \frac{K_1 + \beta [0.5 (1 - \beta^2) + (1 - \beta^2)^2]}{\beta^4}$
 $K_1 = 55 f_T$
 $\beta = 0.9$
- $K = 14 f_T$ 90° weld elbows; page A-29
 $K = f \frac{L}{D}$ pipe; page 3-4
 $R_e = 6.31 \frac{W}{d\mu}$ page 3-2
- $d = 5.761$ 6" Sched. 80 pipe; page B-17
 $\sqrt{V} = 1.216$ 600 psi steam, 850 F; page A-17
 $\mu = 0.027$ page A-2
 $f_T = 0.015$ page A-26
- For globe valve,
 $K_2 = \frac{55 \times 0.015 + 0.9 [0.5 (1 - 0.9^2) + (1 - 0.9^2)^2]}{0.9^4}$
 $K_2 = 1.44$
- $R_e = \frac{6.31 \times 90,000}{5.761 \times 0.027} = 3.65 \times 10^5$
 $f = 0.015$ pipe; page A-25
 $K = \frac{0.015 \times 400 \times 12}{5.761} = 12.5$ pipe
 $K = 3 \times 14 \times 0.015 = 0.63$ 3 elbows; page A-29
 $K_2 = 1.44$ 6 x 4" gate valve; Example 4.4
- Summarizing K for the entire system (globe valve, pipe, venturi gate valve, and elbows),
 $K = 1.44 + 12.5 + 0.63 + 1.44 = 16$
- $\Delta P = \frac{28 \times 10^{-8} \times 16 \times 90^2 \times 1.216}{5.761^4}$
 $\Delta P = 40.1$

Example 4-11 . . . Flat Heating Coils—Water

Given: Water at 180 F is flowing through a flat heating coil, shown in the sketch below, at a rate of 15 gallons per minute.



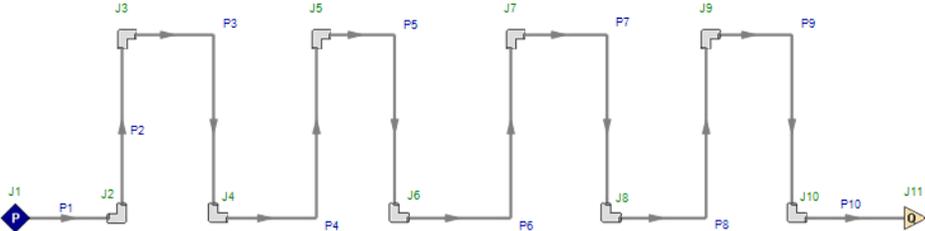
Find: The pressure drop from Point A to B.

Solution:

- $\Delta P = \frac{18 \times 10^{-8} K_D Q^2}{d^5}$ page 3-4
 $R_e = \frac{50.6 Q_D}{d\mu}$ page 3-2
 $K = f \frac{L}{D}$ straight pipe; page 3-4
 $r/d = 4$ pipe bends
 $K_{90} = 14 f_T$ 90° bends; page A-29
 $K_B = (n-1) (25 \pi f_T \frac{r}{d} + .5 K_{90}) + K_{90}$
. 180° bends; page A-29
- $\rho = 60.57$ water, 180 F; page A-6
 $\mu = 0.34$ water, 180 F; page A-3
 $d = 1.049$ 1" Sched. 40 pipe; page B-16
 $f_T = 0.023$ 1" Sched. 40 pipe; page A-26
- $R_e = \frac{50.6 \times 15 \times 60.57}{1.049 \times 0.34} = 1.3 \times 10^5$
 $f = 0.024$ pipe
 $K = \frac{0.024 \times 18 \times 12}{1.049} = 4.94$ 18' straight pipe
 $K = 2 \times 14 \times 0.023 = 0.64$ two 90° bends
- For seven 180° bends,
 $K_B = 7[(2-1) (0.25 \pi \times 0.023 \times 4) + (0.5 \times 0.32) + 0.32] = 3.87$
- $K_{TOTAL} = 4.94 + 0.64 + 3.87 = 9.45$
- $\Delta P = \frac{18 \times 10^{-8} \times 9.45 \times 15^2 \times 60.57 \times 1.216}{1.049^4} = 1.91$

View Verification Case 33 Model

[Verification Case 33](#)



Verification Case 34

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify34.fth

REFERENCE: Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-7, Example 4-13

FLUID: Fuel Oil

ASSUMPTIONS: Inlet pressure is 2 bars

RESULTS:

Parameter	Crane	AFT Fathom
Head loss (meters)	8.95	8.91
Pressure drop (kg/cm ²)	0.729	0.7262
Pressure drop (bar)	0.715	0.7121
Pressure drop (MPa)	0.0715	0.07121

DISCUSSION:

The viscosity is given as kinematic. Converting to dynamic viscosity yields 2.20E-03 kg/m-s.

The head loss in AFT Fathom can be found in three places. First, it is given in the pipe output table. Second, it can be obtained by subtracting the discharge EGL from the supply. Third, it can be viewed in the Junction Deltas summary at the top of the Output window. The junction delta was setup in the Output Control window.

The pressure loss in AFT Fathom can be found in the same three places. Note that three junction deltas are setup for pressure, with different units for each. This allows a more straightforward comparison with the Crane results.

[List of All Verification Models](#)

Verification Case 34 Problem Statement

Verification Case 34

Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-7, Example 4-13

Crane Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Pressure Drop and Velocity in Piping Systems — continued

Example 4-12 ... Orifice Size for Given Pressure Drop and Velocity

Given: A 12-inch Schedule 40 steel pipe 60 feet long, containing a standard gate valve 10 feet from the entrance, discharges 60 F water to atmosphere from a reservoir. The entrance projects inward into the reservoir and its center line is 12 feet below the water level in the reservoir.

Find: The diameter of thin-plate orifice that must be centrally installed in the pipe to restrict the velocity of flow to 10 feet per second when the gate valve is wide open.

Solution:

$$1. \quad h_L = K \frac{v^2}{2g} \text{ or System } K = \frac{2gh_L}{v^2} \quad \dots \text{page 3-4}$$

$$R_e = \frac{123.9 \, dv \rho}{\mu} \quad \dots \text{page 3-2}$$

$$2. \quad K = 0.78 \quad \dots \text{entrance; page A-29}$$

$$K = 1.0 \quad \dots \text{exit; page A-29}$$

$$K_1 = 8f_T \quad \dots \text{gate valve; page A-27}$$

$$K = f \frac{L}{D} \quad \dots \text{pipe; page 3-4}$$

$$3. \quad d = 11.938 \quad \dots \text{pipe; page B-17}$$

$$f_T = 0.013 \quad \dots \text{page A-26}$$

$$\rho = 62.371 \quad \dots \text{page A-6}$$

$$\mu = 1.1 \quad \dots \text{page A-3}$$

$$4. \quad R_e = \frac{123.9 \times 11.938 \times 10 \times 62.371}{1.1} = 8.4 \times 10^5$$

$$f = 0.014 \quad \dots \text{page A-25}$$

$$5. \quad \text{Total } K \text{ required} = 64.4 \times 12 + 10^2 = 7.72$$

$$K_1 = 8 \times 0.013 = 0.10 \quad \dots \text{gate valve}$$

$$K = 60 \times 0.014 = 0.84 \quad \dots \text{pipe}$$

Then, exclusive of orifice,

$$K_{\text{total}} = 0.78 + 1.0 + 0.1 + 0.84 = 2.72$$

$$6. \quad K_{\text{orifice}} = 7.72 - 2.72 = 5$$

$$7. \quad K_{\text{orifice}} \approx \frac{1 - \beta^2}{C^2 \beta^4} \quad \dots \text{page A-20}$$

$$8. \quad \text{Assume } \beta = 0.7 \quad \therefore C = 0.7 \quad \text{page A-20}$$

then $K \approx 4.3 \quad \therefore \beta$ is too large

$$9. \quad \text{Assume } \beta = 0.65 \quad \therefore C = 0.67 \quad \text{page A-20}$$

then $K \approx 7.1 \quad \therefore \beta$ is too small

$$10. \quad \text{Assume } \beta = 0.67 \quad \therefore C = 0.682 \quad \text{page A-20}$$

then $K \approx 5.8 \quad \therefore$ use $\beta = 0.68$

$$11. \quad \text{Orifice size} \approx 11.938 \times 0.68 = 8.1''$$

Example 4-13 ... Flow Given in International Metric System (SI) Units—Oil

Given: Fuel oil with a density of 0.815 grams per cubic centimeter and a kinematic viscosity of 2.7 centistokes is flowing through 50 millimeter I.D. steel pipe (30 meters long) at a rate of 7.0 liters per second.

Find: Head loss in meters of fluid and pressure drop in kg/cm², bar, and megapascal (MPa).

Solution: 1. Define symbols in SI units as follows:

- A ... cross-sectional area of pipe, in meters²
- D ... internal diameter of pipe, in meters
- g ... acceleration of gravity = 9.8 meters/sec/sec
- h_L ... head loss, in meters of fluid
- L ... length of pipe, in meters
- q ... rate of flow, in meters³/second
- v ... mean velocity of flow, in meters/second
- ρ ... fluid density, in grams/centimeter³
- ΔP (kg/cm²) ... pressure drop, in kilograms/centimeter²
- ΔP (bar) ... pressure drop, in bars
- ΔP (MPa) ... pressure drop, in megapascals

2. Use metric-imperial equivalents as indicated below and on pages B-10 and B-11.

$$\text{meter (1)} = 3.28 \text{ feet} = 39.37 \text{ inches}$$

$$\text{bar} = 0.98067 \times \text{kg/cm}^2$$

$$\text{megapascal} = 0.098067 \times \text{kg/cm}^2$$

A column of fluid one square centimeter in cross-sectional area and one meter high is equal to a pressure of 0.1 ρ kg/cm²; therefore:

$$\Delta P \text{ (kg/cm}^2\text{)} \text{ equals } \dots 0.1 \rho h_L$$

$$\Delta P \text{ (bar)} \text{ equals } \dots 0.98067 \Delta P \text{ (kg/cm}^2\text{)}$$

$$\Delta P \text{ (MPa)} \text{ equals } \dots 0.098067 \Delta P \text{ (kg/cm}^2\text{)}$$

$$3. \quad v = \frac{q}{A} = \frac{7 \times 10^{-3}}{(\pi/4) \times 50^2 \times 10^{-6}} = 3.566 \quad \dots \text{page 3-2}$$

$$R_e = \frac{7740 \times 39.37 D \times 3.28 v}{\nu} = \frac{D v \times 10^6}{\nu} \quad \dots \text{page 3-2}$$

$$R_e = \frac{0.050 \times 3.566 \times 10^6}{2.7} = 6.6 \times 10^4$$

$$f = 0.023 \quad \dots \text{page A-25}$$

$$4. \quad h_L = f \frac{L}{D} \frac{v^2}{2g} = \frac{0.023 \times 30 \times 3.566^2}{0.050 \times 2 \times 9.8} = 8.95 \quad \dots \text{page 3-4}$$

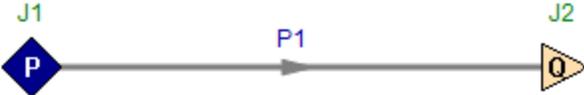
$$\Delta P \text{ (kg/cm}^2\text{)} = 0.1 \times 0.815 \times 8.95 = 0.729$$

$$\Delta P \text{ (bar)} = 0.98067 \times 0.729 = 0.715$$

$$\Delta P \text{ (MPa)} = 0.098067 \times 0.729 = 0.0715$$

View Verification Case 34 Model

[Verification Case 34](#)



Verification Case 35

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify35.fth

REFERENCE: Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-8, Example 4-14

FLUID: Water at 60 deg. F

ASSUMPTIONS: Inlet pressure is 100 psig

RESULTS:

Parameter	Crane	AFT Fathom
Static Head loss (feet)	15.8	15.5
Static Pressure drop (psid)	39.0	39.22

DISCUSSION:

The head loss in AFT Fathom can be found in three places. First, it is given in the pipe output table. Second, it can be obtained by subtracting the discharge HGL from the supply. Third, it can be viewed in the Junction Deltas summary at the top of the Output window. The junction delta was setup in the Output Control window.

The pressure loss in AFT Fathom can be found by subtracting the discharge pressure from the supply. It also can be found in the Junction Deltas on the Output window.

[List of All Verification Models](#)

Verification Case 35 Problem Statement

Verification Case 35

Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-8, Example 4-14

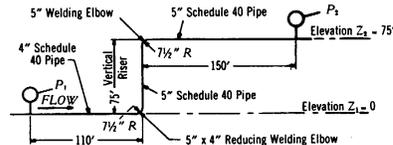
Crane Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Pressure Drop and Velocity in Piping Systems — continued

Example 4-14 . . . Bernoulli's Theorem—Water

Given: Water at 60 F is flowing through the piping system, shown in the sketch below, at a rate of 400 gallons per minute.



Find: The velocity in both the 4 and 5-inch pipe sizes and the pressure differential between gauges P_1 and P_2 .

Solution:

1. Use Bernoulli's theorem (see page 3-2):

$$Z_1 + \frac{1.44 P_1}{\rho_1} + \frac{v_1^2}{2g} = Z_2 + \frac{1.44 P_2}{\rho_2} + \frac{v_2^2}{2g} + h_L$$

Since, $\rho_1 = \rho_2$

$$P_1 - P_2 = \frac{\rho}{1.44} \left((Z_2 - Z_1) + \frac{v_2^2 - v_1^2}{2g} + h_L \right)$$

2. $h_L = \frac{0.00259 K Q^2}{d^5}$ page 3-4

$R_e = \frac{50.6 Q \rho}{d \mu}$ page 3-2

$K = f \frac{L}{D}$ page 3-4

$K = \frac{fL}{D\beta^4}$ (small pipe, in terms of larger pipe; page 2-11)

$K = 14 f_T$ 90° elbow; page A-29

$K = 14 f_T + \frac{(1 - \beta^2)^2}{\beta^4}$ reducing 90° elbow; page A-26

Note: In the absence of test data for increasing elbows, the resistance is conservatively estimated to be equal to the summation of the resistance due to a straight size elbow and a sudden enlargement.

$\beta = \frac{d_1}{d_2}$ page A-26

3. $\rho = 62.371$ page A-6

$\mu = 1.1$ page A-3

$d_1 = 4.026$ 4" Sched. 40 pipe; page B-17

$d_2 = 5.047$ 5" Sched. 40 pipe; page B-17

$f_T = 0.016$ 5" size; page A-26

4. $\beta = \frac{4.026}{5.047} = 0.80$

$Z_2 - Z_1 = 75 - 0 = 75$ feet

$v_1 = 10.08$ 4" pipe; page B-14

$v_2 = 6.42$ 5" pipe; page B-14

$\frac{v_2^2 - v_1^2}{2g} = \frac{6.42^2 - 10.08^2}{2 \times 32.2} = -0.94$ feet

5. For Schedule 40 pipe,

$R_e = \frac{50.6 \times 400 \times 62.371}{4.026 \times 1.1} = 2.85 \times 10^5$ 4" pipe

$R_e = \frac{50.6 \times 400 \times 62.371}{5.047 \times 1.1} = 2.27 \times 10^5$ 5" pipe

$f = 0.018$ 4 or 5" pipe; page A-25

6. $K = \frac{0.018 \times 225 \times 12}{5.047}$, or

$K = 9.6$ for 225' of 5" Sched. 40 pipe

$K = \frac{0.018 \times 110 \times 12}{4.026}$, or

$K = 5.9$ for 110' of 4" Sched. 40 pipe

With reference to velocity in 5" pipe,

$K_2 = 5.9 + 0.8^4 = 14.4$

$K = 14 \times 0.016 = 0.22$ 5" 90° elbow

$K = 0.22 + \frac{0.36^2}{0.8^4} = 0.54$ 5x4" 90° elbow

7. Then, in terms of 5-inch pipe,

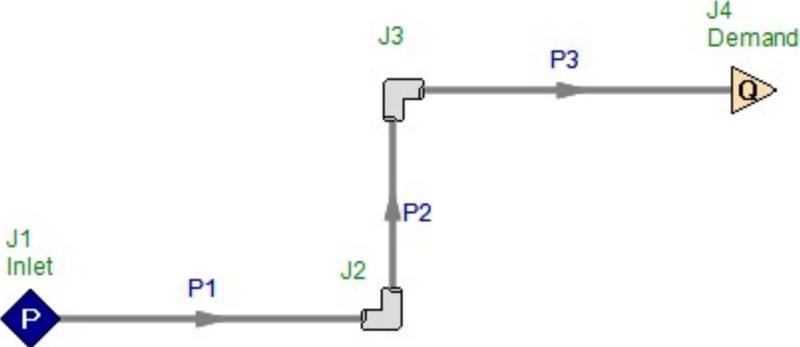
$K_{TOTAL} = 9.6 + 14.4 + 0.22 + 0.54 = 24.8$

8. $h_L = \frac{0.00259 \times 24.8 \times 400^2}{5.047^4} = 15.8$

9. $P_1 - P_2 = \frac{62.371}{144} (75 - 0.94 + 15.8) = 39.0$

View Verification Case 35 Model

[Verification Case 35](#)



Verification Case 36

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify36.fth

REFERENCE: Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-9, Example 4-15

FLUID: Water at 70 deg. F

ASSUMPTIONS: N/A

RESULTS:

Parameter	Crane	AFT Fathom
Head loss (feet)	21	20.5
Pump added head (feet)	421	420.5
Pump power usage @ 70% efficiency (hp)	15.2	15.19

DISCUSSION:

AFT Fathom did not have the globe valve with reducers in its library, so the K factor of 27 was entered directly as a fitting and loss value in pipe 2. The standard gate valve was in the library, and was included in pipe 2. No information was provided about the pump suction, so a reservoir with zero elevation was used as the inlet with a frictionless suction pipe. Also, the discharge at 400 feet was to a reservoir connected to the last elbow by a frictionless pipe. The pump was modeled as an assigned flow.

The head loss from the pump discharge to the discharge tank can be obtained in AFT Fathom by subtracting the reservoir EGL from the pump outlet EGL. It also can be viewed in the Junction Deltas summary at the top of the Output window. The junction delta was setup in the Output Control window.

The added head in AFT Fathom is given in the Pump Summary on the Output window. The ideal power usage is also given in the Pump Summary as 10.62 hp. Dividing by 0.7 for the efficiency obtains 15.18 as shown in the table above.

[List of All Verification Models](#)

Verification Case 36 Problem Statement

Verification Case 36

Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-9, Example 4-15

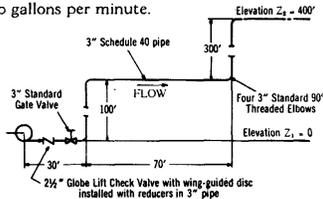
Crane Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Pressure Drop and Velocity in Piping Systems — continued

Example 4-15... Power Required for Pumping

Given: Water at 70 F is pumped through the piping system below at a rate of 100 gallons per minute.



Find: The total discharge head (H) at flowing conditions and the brake horsepower (bhp) required for a pump having an efficiency (e_p) of 70 per cent.

Solution: 1. Use Bernoulli's theorem (see page 3-2):

$$Z_1 + \frac{1.44 P_1}{\rho_1} + \frac{v_1^2}{2g} = Z_2 + \frac{1.44 P_2}{\rho_2} + \frac{v_2^2}{2g} + h_L$$

2. Since $\rho_1 = \rho_2$ and $v_1 = v_2$, the equation can be rewritten to establish the pump head, H :

$$\frac{1.44}{\rho} (P_1 - P_2) = (Z_2 - Z_1) + h_L$$

3. $h_L = \frac{0.00259 K Q^2}{d^5}$ page 3-4

$R_s = 123.9 \frac{d v \rho}{\mu}$ page 3-2

$v = \frac{0.408 Q}{d^2}$ page 3-2

$bhp = \frac{Q H \rho}{247,000 e_p}$ page B-9

4. $K = 30 f_T$ 90° elbow; page A-29

$K_1 = 8 f_T$ gate valve; page A-27

$K = f \frac{L}{D}$ straight pipe; page 3-4

$K = 1.0$ exit; page A-29

5. $d = 3.068$ 3" Sched. 40 pipe; page B-16

$\rho = 62.305$ page A-6

$\mu = 0.95$ page A-3

$f_T = 0.018$ page A-26

6. $v = \frac{0.408 \times 100}{3.068^2} = 4.33$

$R_s = \frac{123.9 \times 3.068 \times 4.33 \times 62.305}{0.95} = 1.1 \times 10^4$

$f = 0.021$ page A-25

7. $K = 4 \times 30 \times 0.018 = 2.16$ four 90° elbows

$K_1 = 8 \times 0.018 = 0.14$ gate valve

$K = 27.0$ lift check valve with reducers; Example 4-5

For 500 feet of 3-inch Schedule 40 pipe,

$$K = \frac{0.021 \times 500 \times 12}{3.068} = 41.06$$

And,

$$K_{TOTAL} = 2.16 + 0.14 + 27.0 + 41.06 + 1 = 71.4$$

8. $h_L = \frac{0.00259 \times 71.4 \times 100^2}{3.068^5} = 21$

9. $H = 400 + 21 = 421$

$$bhp = \frac{100 \times 421 \times 62.305}{24700 \times 0.70} = 15.2$$

Example 4-16... Air Lines

Given: Air at 65 psig and 110 F is flowing through 75 feet of 1-inch Schedule 40 pipe at a rate of 100 standard cubic feet per minute (scfm).

Find: The pressure drop in pounds per square inch and the velocity in feet per minute at both upstream and downstream gauges.

Solution: 1. Referring to the table on page B-15, read pressure drop of 2.21 psi for 100 psi, 60 F air at a flow rate of 100 scfm through 100 feet of 1-inch Schedule 40 pipe.

2. Correction for length, pressure, and temperature (page B-15):

$$\Delta P = 2.21 \left(\frac{75}{100} \right) \left(\frac{100 + 14.7}{65 + 14.7} \right) \left(\frac{460 + 110}{520} \right)$$

$$\Delta P = 2.61$$

3. To find the velocity, the rate of flow in cubic feet per minute at flowing conditions must be determined from page B-15.

$$q_m = q'_m \left(\frac{14.7}{14.7 + P} \right) \left(\frac{460 + t}{520} \right)$$

At upstream gauge:

$$q_m = 100 \left(\frac{14.7}{14.7 + 65} \right) \left(\frac{460 + 110}{520} \right) = 20.2$$

At downstream gauge:

$$q_m = 100 \left[\frac{14.7}{14.7 + (65 - 2.51)} \right] \left(\frac{460 + 110}{520} \right) = 20.9$$

4. $V = \frac{q_m}{A}$ page 3-2

5. $A = 0.0061$ page B-16

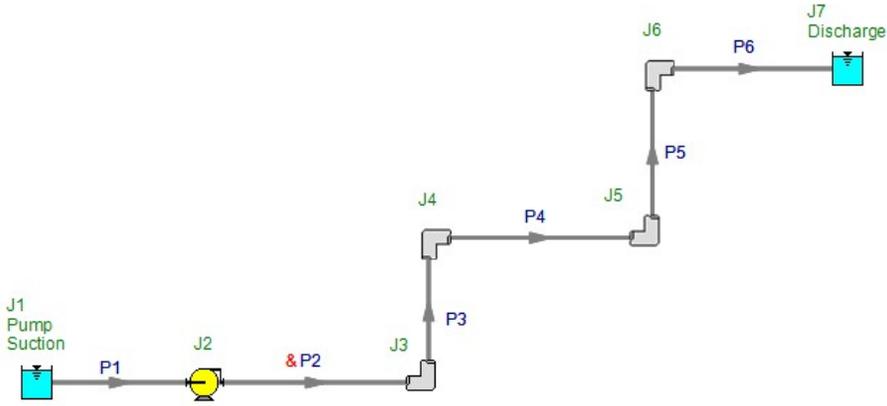
6. $V = \frac{20.2}{0.0061} = 3367$ at upstream gauge

$V = \frac{20.9}{0.0061} = 3483$ at downstream gauge

Note: Example 4-16 may also be solved by use of the pressure drop formula and nomograph shown on pages 3-2 and 3-21 respectively or the velocity formula and nomograph shown on pages 3-2 and 3-17 respectively.

View Verification Case 36 Model

[Verification Case 36](#)



Verification Case 37

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify37.fth

REFERENCE: Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-10, Example 4-17

FLUID: Crude Oil 30 degree API

ASSUMPTIONS: Assumed a rated pump speed of 1800 rpm for the HI viscosity correction calculation.

RESULTS:

Parameter	Crane	AFT Fathom
Head loss (feet)	1,405	1,409
Pump added head (feet)	3,405	3,409
Pump power usage @ 67% efficiency (hp)	1,496	1,498

DISCUSSION:

No information was provided about the pump suction, so a reservoir with zero elevation was used as the inlet with a frictionless suction pipe. The pump was modeled as an assigned flow.

The head loss from the pump discharge to the discharge tank can be obtained in AFT Fathom by subtracting the reservoir EGL from the pump outlet EGL. It also can be viewed in the Junction Deltas summary at the top of the Output window. The junction delta was setup in the Output Control window.

The added head in AFT Fathom is given in the Pump Summary on the Output window. The ideal power usage is also given in the Pump Summary as 1004 hp. Dividing by 0.67 for the efficiency obtains 1498 as shown in the table above.

[List of All Verification Models](#)

Verification Case 37 Problem Statement

[Verification Case 37](#)

Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-10, Example 4-17

[Crane Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Pipe Line Flow Problems

Example 4-17... Sizing of Pump for Oil Pipe Lines

Given: Crude oil 30 degree API at 15.6 C with a viscosity of 75 Universal Saybolt seconds is flowing through a 12-inch Schedule 30 steel pipe at a rate of 1900 barrels per hour. The pipe line is 50 miles long with discharge at an elevation of 2000 feet above the pump inlet. Assume the pump has an efficiency of 67 per cent.

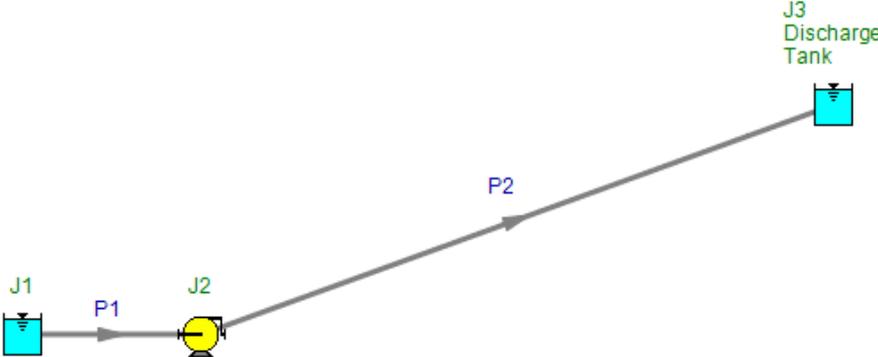
Find: The brake horsepower of the pump.

Solution:

1. $\Delta P = 0.0001058 \frac{fL\rho B^2}{d^5}$ {Equation 3-5 on page 3-2
or, after converting B to Q ,
use nomograph on page 3-11
- $t = 1.8 t_c + 32$ page B-10
- $R_s = 35.4 \frac{B\rho}{d\mu}$ page 3-2 or 3-8
- $h_L = \frac{144\Delta P}{\rho}$ page 3-5
- brake horsepower = $\frac{QH\rho}{247000e_p}$ page B-9
2. $t = (1.8 \times 15.6) + 32 = 60$ F
3. $\rho = 54.64$ page B-7
 $S = 0.8762$ page B-7
4. $d = 12.09$ page B-17
 $d^5 = 258304$
5. 75 USS = 12.5 centipoise page B-5
6. $R_s = \frac{35.4 \times 1900 \times 54.64}{12.09 \times 12.5} = 24300$
7. $f = 0.025$ page A-25
8. $\Delta P = \frac{0.0001058 \times 0.025 \times 50 \times 2580 \times 54.64 \times 1900^2}{258304}$
 $\Delta P = 533$
9. $h_L = \frac{144 \times 533}{54.64} = 1405$
10. The total discharge head at the pump is:
 $H = 1405 + 2000 = 3405$
11. $Q = \left(\frac{1900 \text{ bbl}}{\text{hr}}\right) \left(\frac{42 \text{ gal}}{\text{bbl}}\right) \left(\frac{\text{hr}}{60 \text{ min}}\right) = 1330$
12. Then, the brake horsepower is:
 $\frac{1330 \times 3405 \times 54.64}{247000 \times 0.67} = 1496$, or say 1500

View Verification Case 37 Model

[Verification Case 37](#)



Verification Case 38

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify38.fth

REFERENCE: Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-12, Example 4-19

FLUID: Water at 60 deg. F

ASSUMPTIONS: N/A

RESULTS:

Parameter	Crane	AFT Fathom
Flow rate (gpm)	137	139

DISCUSSION:

Crane used friction factors of 0.021 and 0.020 based on a chart lookup. AFT Fathom calculated friction factors of 0.0201 and 0.0206 for the two pipes based on the Colebrook-White correlation. Crane's slightly larger friction factor from the chart lookup is the reason the results differ slightly.

[List of All Verification Models](#)

Verification Case 38 Problem Statement

Verification Case 38

Crane Co., Flow of Fluids Through Valves, Fittings, and Pipe, Technical Paper No. 410, Crane Co., Joliet, IL, 1988, Page 4-12, Example 4-19

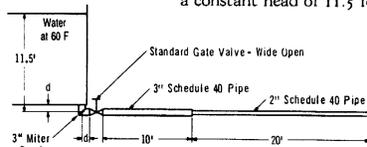
Crane Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

Discharge of Fluids from Piping Systems

Example 4-19... Water

Given: Water at 60 F is flowing from a reservoir through the piping system below. The reservoir has a constant head of 11.5 feet.



Find: The flow rate in gallons per minute.

Solution: $Q = 19.65 \text{ d}^2 \sqrt{\frac{h_L}{K}}$... page 3-4

$$R_e = \frac{50.6 Q_p}{d\mu} \dots \text{page 3-2}$$

$$\beta = d_1/d_2 \dots \text{page A-26}$$

$$K = 0.5 \dots \text{entrance; page A-29}$$

$$K = 60 f_r \dots \text{mitre bend; page A-29}$$

$$K_1 = 8 f_r \dots \text{gate valve; page A-27}$$

$$K = f \frac{L}{D} \dots \text{straight pipe; page 3-4}$$

$$K_2 = \frac{0.5 (1 - \beta^2) \sqrt{\sin \frac{\theta}{2}}}{\beta^4} \dots \text{sudden contraction; page A-26}$$

$$K = \frac{fL}{D\beta^4} \dots \left\{ \begin{array}{l} \text{small pipe, in terms of} \\ \text{larger pipe; page 2-5} \end{array} \right.$$

$$K = \frac{1}{\beta^4} \dots \left\{ \begin{array}{l} \text{exit from small pipe} \\ \text{in terms of larger pipe} \end{array} \right.$$

$$3. \quad d = 2.067 \dots \text{2" Sched. 40 pipe; page B-16}$$

$$d = 3.068 \dots \text{3" Sched. 40 pipe; page B-16}$$

$$\mu = 1.1 \dots \text{page A-3}$$

$$\rho = 62.371 \dots \text{page A-6}$$

$$f_r = 0.019 \dots \text{2" pipe; page A-26}$$

$$f_r = 0.018 \dots \text{3" pipe; page A-26}$$

$$4. \quad \beta = 2.067 / 3.068 = 0.67$$

$$K = 0.5 \dots \text{3" entrance}$$

$$K = 60 \times 0.018 = 1.08 \dots \text{3" mitre bend}$$

$$K_1 = 8 \times 0.018 = 0.14 \dots \text{3" gate valve}$$

$$K = \frac{0.018 \times 10 \times 12}{3.068} = 0.70 \dots \text{10 feet, 3" pipe}$$

For 20 feet of 2-inch pipe, in terms of 3-inch pipe,

$$K = \frac{0.019 \times 20 \times 12}{2.067 \times 0.67^4} = 10.9$$

For 2-inch exit, in terms of 3-inch pipe,

$$K = 1 + 0.67^4 = 5.0$$

For sudden contraction,

$$K_2 = \frac{0.5 (1 - 0.67^2) (1)}{0.67^4} = 1.37$$

$$\text{and, } K_{\text{TOTAL}} = 0.5 + 1.08 + 0.14 + 0.70 + 10.9 + 5.0 + 1.37 = 19.7$$

$$5. \quad Q = 19.65 \times 3.068^2 \sqrt{11.5 + 19.7} = 141$$

(this solution assumes flow in fully turbulent zone)

6. Calculate Reynolds numbers and check friction factors for flow in straight pipe of the 2-inch size:

$$R_e = \frac{50.6 \times 141 \times 62.371}{2.067 \times 1.1} = 1.96 \times 10^5$$

$$f = 0.021 \dots \text{page A-25}$$

and for flow in straight pipe of the 3-inch size:

$$R_e = \frac{50.6 \times 141 \times 62.371}{3.068 \times 1.1} = 1.32 \times 10^5$$

$$f = 0.020 \dots \text{page A-25}$$

7. Since assumed friction factors used for straight pipe in Step 4 are not in agreement with those based on the approximate flow rate, the K factors for these items and the total system should be corrected accordingly.

$$K = \frac{0.020 \times 10 \times 12}{3.068} = 0.78 \dots \text{10 feet, 3" pipe}$$

For 20 feet of 2-inch pipe, in terms of 3-inch pipe,

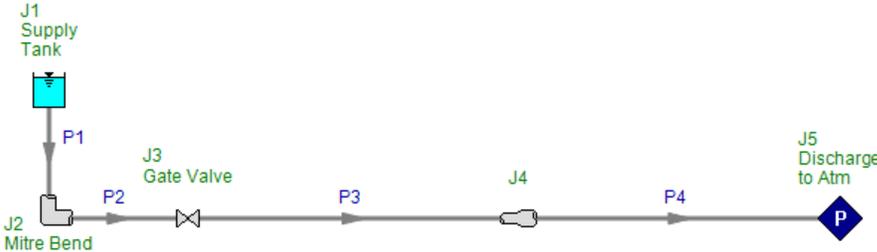
$$K = \frac{0.021 \times 20 \times 12}{2.067 \times 0.67^4} = 12.1$$

$$\text{and, } K_{\text{TOTAL}} = 0.5 + 1.08 + 0.14 + 0.78 + 12.1 + 5.0 + 1.37 = 21.0$$

$$8. \quad Q = 19.65 \times 3.068^2 \sqrt{11.5 + 21} = 137$$

View Verification Case 38 Model

[Verification Case 38](#)



Verification Case 39

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify39.fth

REFERENCE: Robert W. Fox and Alan T. McDonald, Introduction to Fluid Mechanics, 3rd Ed., 1985, John Wiley & Sons, Page 373-374, example 8.5

FLUID: Water at 20 deg. C

ASSUMPTIONS: N/A

RESULTS:

Parameter	Fox & McDonald	AFT Fathom
EGL inlet (meters)	44.6	44.702

DISCUSSION:

The supply reservoir is modeled as an Assigned Flow junction at $0.03 \text{ m}^3/\text{sec}$. The solution for inlet EGL is the reservoir height required.

[List of All Verification Models](#)

Verification Case 39 Problem Statement

Verification Case 39

Robert W. Fox and Alan T. McDonald, Introduction to Fluid Mechanics, 3rd Ed., 1985, John Wiley & Sons, Page 373-374, example 8.5

Fox and McDonald Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

SOLUTION OF PIPE FLOW PROBLEMS/8-8 **373**

friction factor is obtained from Fig. 8.14. Then the head loss is computed from Eqs. 8.32 and 8.38, and Eq. 8.28 is solved for pressure drop. The resulting trial value is compared to the system requirement.

If the trial value of Δp is too large, calculations are repeated for a larger assumed value of D . If the trial value of Δp is less than the criterion, a smaller assumed value of D should be checked.

In choosing a pipe diameter, it is logical to work with values that are available commercially. Pipe is manufactured in a limited number of standard sizes. Some data for standard pipe sizes are given in Table 8.4. For data on extra strong or double extra strong pipes, consult a handbook, e.g. [7]. Pipe larger than 12 in. nominal diameter is produced in multiples of 2 in. up to a nominal diameter of 36 in. and in multiples of 6 in. for still larger sizes.

Table 8.4 Standard Sizes for Carbon Steel, Alloy Steel, and Stainless Steel Pipe (Data from [7])

Nominal Pipe Size (in.)	Inside Diameter (in.)	Nominal Pipe Size (in.)	Inside Diameter (in.)
$\frac{1}{8}$	0.269	$2\frac{1}{2}$	2.469
$\frac{1}{4}$	0.364	3	3.068
$\frac{3}{8}$	0.493	4	4.026
$\frac{1}{2}$	0.622	5	5.047
$\frac{3}{4}$	0.824	6	6.065
1	1.049	8	7.981
$1\frac{1}{2}$	1.610	10	10.020
2	2.067	12	12.000

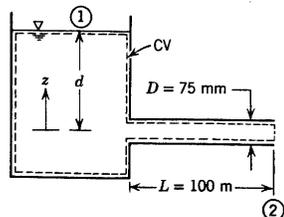
Example 8.5

A 100 m length of smooth horizontal pipe is attached to a large reservoir. What depth, d , must be maintained in the reservoir to produce a volume flow rate of $0.03 \text{ m}^3/\text{sec}$ of water? The inside diameter of the smooth pipe is 75 mm. The loss coefficient, K , for the square-edged inlet is 0.5. The water discharges to the atmosphere.

EXAMPLE PROBLEM 8.5

GIVEN: Water flow at $0.03 \text{ m}^3/\text{sec}$ through a 75 mm diameter pipe, with $L = 100 \text{ m}$, attached to a constant-level reservoir. Inlet loss coefficient, $K = 0.5$.

FIND: Reservoir depth, d , to maintain the flow.



374 8/INTERNAL INCOMPRESSIBLE VISCOUS FLOW

SOLUTION:

Computing equation:

$$\left(\frac{p_1}{\rho} + \alpha_1 \frac{\bar{V}_1^2}{2} + gz_1\right) - \left(\frac{p_2}{\rho} + \alpha_2 \frac{\bar{V}_2^2}{2} + gz_2\right) = h_{i_T} = h_i + h_{i_m} \quad (8.28)$$

where

$$h_i = f \frac{L}{D} \frac{\bar{V}^2}{2} \quad \text{and} \quad h_{i_m} = K \frac{\bar{V}^2}{2}$$

For the given problem, $p_1 = p_2 = p_{\text{atm}}$, $\bar{V}_1 \approx 0$, $\bar{V}_2 = \bar{V}$, and $\alpha_2 \approx 1.0$. If it is assumed that $z_2 = 0$, then $z_1 = d$. Simplifying Eq. 8.28 gives

$$gd - \frac{\bar{V}^2}{2} = f \frac{L}{D} \frac{\bar{V}^2}{2} + K \frac{\bar{V}^2}{2}$$

Then

$$d = \frac{1}{g} \left[f \frac{L}{D} \frac{\bar{V}^2}{2} + K \frac{\bar{V}^2}{2} + \frac{\bar{V}^2}{2} \right] = \frac{\bar{V}^2}{2g} \left[f \frac{L}{D} + K + 1 \right]$$

Since $\bar{V} = \frac{Q}{A} = \frac{4Q}{\pi D^2}$, then

$$d = \frac{8Q^2}{\pi^2 D^4 g} \left[f \frac{L}{D} + K + 1 \right]$$

Assuming water at 20 C, $\rho = 999 \text{ kg/m}^3$, and $\mu = 1.0 \times 10^{-3} \text{ kg/m} \cdot \text{sec}$. Thus

$$Re = \frac{\rho \bar{V} D}{\mu} = \frac{4\rho Q}{\pi \mu D}$$

$$Re = \frac{4}{\pi} \times \frac{999 \text{ kg}}{\text{m}^3} \times \frac{0.03 \text{ m}^3}{\text{sec}} \times \frac{\text{m} \cdot \text{sec}}{1.0 \times 10^{-3} \text{ kg}} \times \frac{1}{0.075 \text{ m}} = 5.09 \times 10^5$$

For smooth pipe, from Fig. 8.14, $f = 0.0131$. Then

$$\begin{aligned} d &= \frac{8Q^2}{\pi^2 D^4 g} \left[f \frac{L}{D} + K + 1 \right] \\ &= \frac{8}{\pi^2} \times \frac{(0.03)^2 \text{ m}^6}{\text{sec}^2} \times \frac{1}{(0.075)^4 \text{ m}^4} \times \frac{\text{sec}^2}{9.81 \text{ m}} \left[(0.0131) \frac{100 \text{ m}}{0.075 \text{ m}} + 0.5 + 1 \right] \\ d &= 44.6 \text{ m} \end{aligned}$$

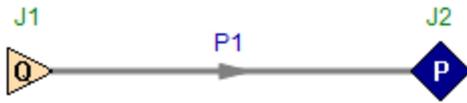
{This problem illustrates the method for calculating the total head loss.}

Example 8.6

A compressed air drill requires an air supply of 0.25 kg/sec at a gage pressure of 650 kPa at the drill. The hose from the air compressor to the drill is 40 mm inside diameter. The maximum compressor discharge gage pressure is 690 kPa. Neglect changes in density and any effects due to hose curvature. Air leaves the compressor at 40 C. Calculate the longest hose that may be used.

View Verification Case 39 Model

[Verification Case 39](#)



Verification Case 40

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify40.fth

REFERENCE: Robert W. Fox and Alan T. McDonald, Introduction to Fluid Mechanics, 3rd Ed., 1985, John Wiley & Sons, Page 376-377, example 8.7

FLUID: Water at 68 deg. F

ASSUMPTIONS: N/A

RESULTS:

Parameter	Fox & McDonald	AFT Fathom
Flow rate (gpm)	350	351.8

DISCUSSION:

There is a slight difference in flow rates because Fox & McDonald use a chart for friction factor, which is less precise than AFT Fathom's correlation based method.

[List of All Verification Models](#)

Verification Case 40 Problem Statement

Verification Case 40

Robert W. Fox and Alan T. McDonald, Introduction to Fluid Mechanics, 3rd Ed., 1985, John Wiley & Sons, Page 376-377, example 8.7

Fox and McDonald Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

376 8/INTERNAL INCOMPRESSIBLE VISCOUS FLOW

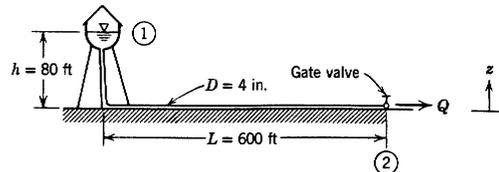
Example 8.7

A fire protection system is supplied from a water tower and standpipe 80 ft tall. The longest pipe in the system is 600 ft long, and is made of cast iron about 20 years old. The pipe contains one gate valve; other minor losses may be neglected. The pipe diameter is 4 in. Determine the maximum rate of flow through this pipe, in gallons per minute.

EXAMPLE PROBLEM 8.7

GIVEN: Fire protection system, as shown.

FIND: Q , gpm.



SOLUTION:

$$\text{Computing equations: } \left(\frac{p_1}{\rho} + \alpha_1 \frac{\bar{V}_1^2}{2} + gz_1 \right) - \left(\frac{p_2}{\rho} + \alpha_2 \frac{\bar{V}_2^2}{2} + gz_2 \right) = h_{l_T} \quad (8.28)$$

$$h_{l_T} = f \frac{L}{D} \frac{\bar{V}_2^2}{2} + h_{l_m} = f \frac{(L + L_e)}{D} \frac{\bar{V}_2^2}{2}$$

Assumptions: (1) $p_1 = p_2 = p_{\text{atm}}$
 (2) $\bar{V}_1 \approx 0$, and $\alpha_2 \approx 1.0$

For a fully open gate valve, from Table 8.3, $L_e/D = 8$. Then

$$h_{l_T} = f \frac{L}{D} \frac{\bar{V}_2^2}{2} + 8f \frac{\bar{V}_2^2}{2} = g(z_1 - z_2) - \frac{\bar{V}_2^2}{2}$$

or

$$\frac{\bar{V}_2^2}{2} \left[f \left(\frac{L}{D} + 8 \right) + 1 \right] = g(z_1 - z_2)$$

Solving for \bar{V}_2 , we obtain

$$\bar{V}_2 = \left[\frac{2g(z_1 - z_2)}{f(L/D + 8) + 1} \right]^{1/2}$$

To be conservative, assume the standpipe is the same diameter as the horizontal pipe. Then

$$\frac{L}{D} = \frac{600 \text{ ft} + 80 \text{ ft}}{4 \text{ in.}} \times \frac{12 \text{ in.}}{\text{ft}} = 2040$$

Also

$$z_1 - z_2 = h = 80 \text{ ft}$$

Since \bar{V}_2 is not known, we cannot compute Re . But we can assume a value of friction factor in the fully rough flow region. From Fig. 8.15, $e/D \approx 0.0025$ for cast iron pipe. Since the pipe is

quite old, choose $e/D = 0.005$. Then, from Fig. 8.14, guess $f \approx 0.03$. Then a first approximation to \bar{V}_2 is

$$\bar{V}_2 = \left[2 \times \frac{32.2 \text{ ft}}{\text{sec}^2} \times 80 \text{ ft} \times \frac{1}{0.03(2040 + 8) + 1} \right]^{1/2} = 9.08 \text{ ft/sec}$$

Now check the value assumed for f .

$$Re = \frac{\rho \bar{V} D}{\mu} = \frac{\bar{V} D}{\nu} = \frac{9.08 \text{ ft}}{\text{sec}} \times \frac{\text{ft}}{3} \times \frac{\text{sec}}{1.2 \times 10^{-5} \text{ ft}^2} = 2.52 \times 10^5$$

For $e/D = 0.005$, $f = 0.031$ from Fig. 8.14. Using this value, we obtain

$$\bar{V}_2 = \left[2 \times \frac{32.2 \text{ ft}}{\text{sec}^2} \times 80 \text{ ft} \times \frac{1}{0.031(2040 + 8) + 1} \right]^{1/2} = 8.94 \text{ ft/sec}$$

Thus convergence is satisfactory. The volume flow rate is

$$Q = \bar{V}_2 A = \bar{V}_2 \frac{\pi D^2}{4} = \frac{8.94 \text{ ft}}{\text{sec}} \times \frac{\pi \left(\frac{1}{3}\right)^2}{4} \text{ ft}^2 \times \frac{7.48 \text{ gal}}{\text{ft}^3} \times \frac{60 \text{ sec}}{\text{min}}$$

$$Q = 350 \text{ gpm}$$

Q

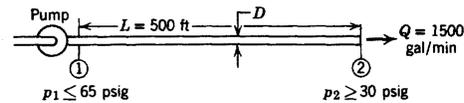
This problem illustrates the procedure for solving pipe flow problems in which the flow rate is unknown. Note that the velocity and, hence, the flow rate, is essentially proportional to $1/\sqrt{f}$. Doubling the value of e/D to account for aging reduced the flow rate by about 10 percent.

Example 8.8

Spray heads in an agricultural spraying system are to be supplied with water through 500 ft of drawn aluminum tubing from an engine-driven pump. In its most efficient operating range, the pump output is 1500 gpm at a discharge pressure not exceeding 65 psig. For satisfactory operation, the sprinklers must operate at 30 psig or higher pressure. Minor losses and elevation changes may be neglected. Determine the smallest standard pipe size that can be used.

EXAMPLE PROBLEM 8.8

GIVEN: Water supply system, as shown.



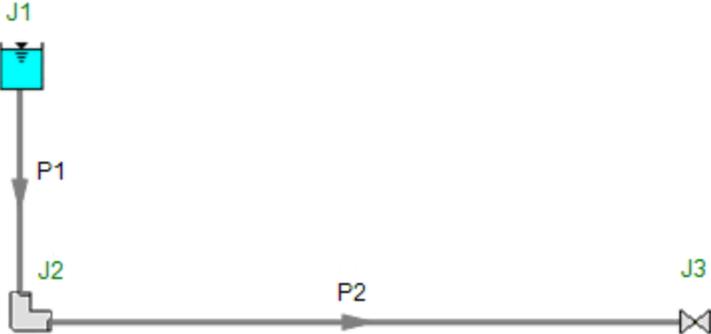
FIND: Smallest standard D .

SOLUTION:

Δp , L , and Q are known. D is unknown, so iteration will be required to determine the minimum standard diameter that satisfies the pressure drop constraint at the given flow rate.

View Verification Case 40 Model

[Verification Case 40](#)



Verification Case 41

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify41.fth

REFERENCE: Robert W. Fox and Alan T. McDonald, Introduction to Fluid Mechanics, 3rd Ed., 1985, John Wiley & Sons, Page 385-389, example 8.11

FLUID: Water at 60 deg. F

ASSUMPTIONS: N/A

RESULTS:

Parameter	Fox & McDonald	AFT Fathom
Flow rate to Branch 1 (gal/min)	493	493.48
Flow rate to Branch 2 (gal/min)	557	557.44
Flow rate to Branch 3 (gal/min)	449	449.09
Pressure at inlet (psig)	91.2	91.52

DISCUSSION:

Fox and McDonald use equivalent lengths to model the elbows. These were converted to K factors of 0.4 in AFT Fathom. The loss factors at the three sprays (modeled as assigned pressure junctions) were specified as 16. This allows for the area ratio of 2 to the 4th power to account for the energy lost at the discharge.

[List of All Verification Models](#)

Verification Case 41 Problem Statement

Verification Case 41

Robert W. Fox and Alan T. McDonald, Introduction to Fluid Mechanics, 3rd Ed., 1985, John Wiley & Sons, Page 385-389, example 8.11

Fox and McDonald Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

SOLUTION OF PIPE FLOW PROBLEMS/8-8 385

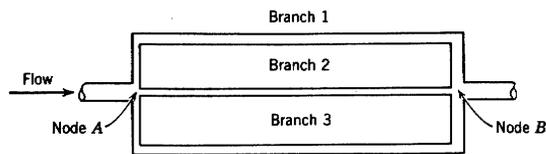
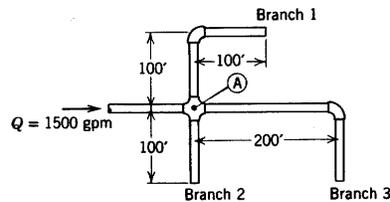


Fig. 8.20 Simple multiple-path pipe flow system.

The fluid flow rate and pressure drop are, respectively, analogous to the current and voltage in an electric circuit. However, the simple linear relation between voltage and current given by Ohm's law does not apply to the fluid flow system. Instead, the flow pressure drop is approximately proportional to the square of the flow rate. This nonlinearity makes iterative solutions necessary, and the resulting calculations can be quite lengthy and tedious. A number of schemes have been developed for use with digital computers. For an example, see [20].

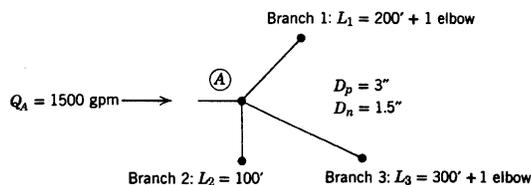
Example 8.11

The irrigation system of Example 8.8 is to be extended, holding the total flow rate constant at 1500 gpm, by adding three branches, each made from 3 in. pipe, as shown in the sketch. The sprinkler at the end of each branch has a nozzle of 1.5 in. minimum diameter. Minor losses at elbows should be included, but elevation terms may be neglected. Determine the flow rate in each branch, the pressure required at (A), and the pressure applied to each sprinkler nozzle.



EXAMPLE PROBLEM 8.11

GIVEN: Pipe network shown schematically in the following figure.



386 8/INTERNAL INCOMPRESSIBLE VISCOUS FLOW

FIND: (a) Q_1, Q_2, Q_3 .

 (b) p_A .

 (c) p_1, p_2, p_3 at nozzle inlets.

SOLUTION:

 Apply Eq. 8.28 to each branch. Choose subscript n for nozzle exit, subscript p for pipe.

Computing equations:

$$\frac{p_A}{\rho} + \alpha_A \frac{\bar{v}_A^2}{2} + g z_A = \frac{p}{\rho} + \alpha_n \frac{\bar{v}_n^2}{2} + g z_n + h_{lr} \quad h_{lr} = f \frac{L}{D} \frac{\bar{v}_p^2}{2} + f \frac{L_e}{D} \frac{\bar{v}_p^2}{2}$$

 Assumptions: (1) $\bar{v}_A^2 \ll \bar{v}_n^2$; $\alpha_n \simeq 1.0$

 (2) $z_A \simeq z_n$

 (3) $p_n = p_{\text{atmosphere}}$

(4) Neglect loss due to flow split at (A).

Then, for each branch of the flow system, using gage pressure at (A), we obtain

$$\frac{p_A}{\rho} = \frac{\bar{v}_n^2}{2} + f \frac{L}{D} \frac{\bar{v}_p^2}{2} + f \frac{L_e}{D} \frac{\bar{v}_p^2}{2}$$

 But for incompressible flow, $\bar{v}_p A_p = \bar{v}_n A_n$, so $\bar{v}_n^2 = \bar{v}_p^2 (A_p/A_n)^2 = \bar{v}_p^2 (D_p/D_n)^4$, and

$$\frac{p_A}{\rho} = \frac{\bar{v}_p^2}{2} \left[\left(\frac{D_p}{D_n} \right)^4 + f \left(\frac{L}{D} + \frac{L_e}{D} \right) \right]$$

or

$$\bar{v}_p = \sqrt{\frac{2p_A}{\rho} \left[\frac{1}{\left(\frac{D_p}{D_n} \right)^4 + f \left(\frac{L}{D} + \frac{L_e}{D} \right)} \right]^{1/2}}$$

The corresponding flow rate for each branch is

$$Q_p = \bar{v}_p A_p = A_p \sqrt{\frac{2p_A}{\rho} \left[\frac{1}{\left(\frac{D_p}{D_n} \right)^4 + f \left(\frac{L}{D} + \frac{L_e}{D} \right)} \right]^{1/2}} \quad (1)$$

 To find f , we must determine the pipe Reynolds number. If flow is equally split as a first approximation, then $Q_p \simeq 500$ gpm per branch, and

$$\begin{aligned} Re &= \frac{\bar{v}_p D_p}{\nu} = \frac{4Q_p}{\pi \nu D_p} \\ &= \frac{4}{\pi} \times \frac{500 \text{ gal}}{\text{min}} \times \frac{\text{sec}}{1.2 \times 10^{-5} \text{ ft}^2} \times \frac{1}{3 \text{ in.}} \times \frac{\text{min}}{60 \text{ sec}} \times \frac{\text{ft}^3}{7.48 \text{ gal}} \times \frac{12 \text{ in.}}{\text{ft}} \\ Re &= 4.73 \times 10^5 \end{aligned}$$

 From Fig. 8.14, for smooth pipe, $f \simeq 0.0133$. For an elbow, $L_e/D = 30$, from Table 8.3. Using these values, we can obtain a first approximation for Q_p for each branch. Substituting into

Eq. 1 yields

$$Q_{p_1} \approx A_p \sqrt{\frac{2p_A}{\rho}} \left[\frac{1}{\left(\frac{3.0}{1.5}\right)^4 + 0.0133 \left(200 \text{ ft} \times \frac{1}{3 \text{ in.}} \times \frac{12 \text{ in.}}{\text{ft}} + 30\right)} \right]^{1/2}$$

$$Q_{p_1} \approx 0.192 A_p \sqrt{\frac{2p_A}{\rho}}$$

$$Q_{p_2} \approx A_p \sqrt{\frac{2p_A}{\rho}} \left[\frac{1}{\left(\frac{3.0}{1.5}\right)^4 + 0.0133 \left(100 \text{ ft} \times \frac{1}{3 \text{ in.}} \times \frac{12 \text{ in.}}{\text{ft}}\right)} \right]^{1/2}$$

$$Q_{p_2} \approx 0.217 A_p \sqrt{\frac{2p_A}{\rho}}$$

and

$$Q_{p_3} \approx A_p \sqrt{\frac{2p_A}{\rho}} \left[\frac{1}{\left(\frac{3.0}{1.5}\right)^4 + 0.0133 \left(300 \text{ ft} \times \frac{1}{3 \text{ in.}} \times \frac{12 \text{ in.}}{\text{ft}} + 30\right)} \right]^{1/2}$$

$$Q_{p_3} \approx 0.176 A_p \sqrt{\frac{2p_A}{\rho}}$$

From continuity,

$$Q_A = Q_{p_1} + Q_{p_2} + Q_{p_3} \approx (0.192 + 0.217 + 0.176) A_p \sqrt{\frac{2p_A}{\rho}} \approx 0.585 A_p \sqrt{\frac{2p_A}{\rho}}$$

Thus

$$\frac{Q_{p_1}}{Q_A} \approx \frac{0.192 A_p \sqrt{\frac{2p_A}{\rho}}}{0.585 A_p \sqrt{\frac{2p_A}{\rho}}} = 0.328$$

or

$$Q_{p_1} \approx (0.328) 1500 \text{ gpm} = 492 \text{ gpm}$$

Similarly

$$\frac{Q_{p_2}}{Q_A} \approx \frac{0.217}{0.585} = 0.371; \quad Q_{p_2} \approx 557 \text{ gpm}$$

$$\frac{Q_{p_3}}{Q_A} \approx \frac{0.176}{0.585} = 0.301; \quad Q_{p_3} \approx 452 \text{ gpm}$$

Better approximations for the Reynolds number and friction factor for each branch now may be calculated.

$$Re_1 = \frac{Q_{p_1}}{500 \text{ gpm}} \times 4.73 \times 10^5 \approx \frac{492}{500} \times 4.73 \times 10^5 = 4.65 \times 10^5$$

Verification Case 41 Problem Statement

8. INTERNAL INCOMPRESSIBLE VISCOUS FLOW

From Fig. 8.14, $f_1 \approx 0.0133$. Similarly,

$$Re_2 \approx \frac{557}{500} \times 4.73 \times 10^5 = 5.27 \times 10^5; \quad f_2 \approx 0.0130$$

and

$$Re_3 \approx \frac{452}{500} \times 4.73 \times 10^5 = 4.28 \times 10^5; \quad f_3 \approx 0.0136$$

Substituting these values into Eq. 1, we obtain

$$Q_{p_1} \approx 0.192 A_p \sqrt{\frac{2p_A}{\rho}}$$

$$Q_{p_2} \approx 0.217 A_p \sqrt{\frac{2p_A}{\rho}}$$

and

$$Q_{p_3} \approx 0.175 A_p \sqrt{\frac{2p_A}{\rho}}$$

From continuity,

$$Q_A = Q_{p_1} + Q_{p_2} + Q_{p_3} \approx (0.192 + 0.217 + 0.175) A_p \sqrt{\frac{2p_A}{\rho}} \approx 0.584 A_p \sqrt{\frac{2p_A}{\rho}}$$

Solving for the individual flow rates,

$$Q_{p_1} \approx \frac{0.192}{0.584} \times 1500 \text{ gpm} = 493 \text{ gpm}$$

$$Q_{p_2} \approx \frac{0.217}{0.584} \times 1500 \text{ gpm} = 557 \text{ gpm}$$

and

$$Q_{p_3} \approx \frac{0.175}{0.584} \times 1500 \text{ gpm} = 449 \text{ gpm} \quad \leftarrow Q_1, Q_2, Q_3$$

Solving Eq. 1 for p_A gives

$$p_A = \frac{\rho}{2} \left(\frac{Q_p}{A_p} \right)^2 \left[\left(\frac{D_p}{D_n} \right)^4 + f \left(\frac{L}{D} + \frac{L_e}{D} \right) \right]$$

Substituting values for Branch ①,

$$p_A = \frac{1}{2} \times \frac{1.94 \text{ slug}}{\text{ft}^3} \left(\frac{493 \text{ gal}}{\text{min}} \times \frac{4}{\pi (3)^2 \text{ in.}^2} \times \frac{\text{ft}^3}{7.48 \text{ gal}} \times \frac{\text{min}}{60 \text{ sec}} \times \frac{144 \text{ in.}^2}{\text{ft}^2} \right)^2$$

$$\times \left[\left(\frac{3.0}{1.5} \right)^4 + 0.0133 \left(200 \text{ ft} \times \frac{1}{3 \text{ in.}} \times \frac{12 \text{ in.}}{\text{ft}} + 30 \right) \right] \frac{\text{lbf} \cdot \text{sec}^2}{\text{slug} \cdot \text{ft}} \times \frac{\text{ft}^2}{144 \text{ in.}^2}$$

$$p_A = 91.2 \text{ lbf/in.}^2$$

The values for Branches ② and ③ by similar calculations are 91.4 and 90.5 psig, respectively. (The slight differences are due to rounding the flow rate values.) Thus

$$p_A \approx 91.0 \text{ psig} \longleftarrow p_A$$

The pressure at the inlet to each sprinkler nozzle may be calculated from the energy equation. The equation between ① and the nozzle inlet, section ②, is

$$\frac{p_A}{\rho} + \frac{\bar{V}_A^2}{2} + gz_A = \frac{p_i}{\rho} + \frac{\bar{V}_p^2}{2} + gz_i + h_{i_r} \quad h_{i_r} = f \frac{L}{D} \frac{\bar{V}_p^2}{2} + f \frac{L_e}{D} \frac{\bar{V}_p^2}{2}$$

From continuity, $\bar{V}_A = Q_A/A_A$ and $\bar{V}_p = Q_p/A_p$, so

$$p_i = p_A + \frac{\rho}{2} \left[\left(\frac{Q_A}{A_A} \right)^2 - \left(f \frac{L}{D} + f \frac{L_e}{D} + 1 \right) \left(\frac{Q_p}{A_p} \right)^2 \right]$$

or

$$p_i = p_A + \frac{\rho}{2} \left(\frac{Q_A}{A_A} \right)^2 \left\{ 1 - \left[f \left(\frac{L}{D} + \frac{L_e}{D} \right) + 1 \right] \left(\frac{Q_p}{Q_A} \right)^2 \left(\frac{A_A}{A_p} \right)^2 \right\}$$

Substituting values for Branch ① gives

$$p_{i_1} = \frac{91.0 \text{ lbf}}{\text{in.}^2} + \frac{1}{2} \times \frac{1.94 \text{ slug}}{\text{ft}^3} \left(\frac{1500 \text{ gal}}{\text{min}} \times \frac{4}{\pi (6)^2 \text{ in.}^2} \times \frac{\text{ft}^3}{7.48 \text{ gal}} \times \frac{\text{min}}{60 \text{ sec}} \times \frac{144 \text{ in.}^2}{\text{ft}^2} \right)^2$$

$$\times \left\{ 1 - \left[0.0133 \left(200 \text{ ft} \times \frac{1}{3 \text{ in.}} \times \frac{12 \text{ in.}}{\text{ft}} + 30 \right) + 1 \right] \left(\frac{493}{1500} \right)^2 \left(\frac{6}{3} \right)^4 \right\} \frac{\text{lbf} \cdot \text{sec}^2}{\text{slug} \cdot \text{ft}}$$

$$\times \frac{\text{ft}^2}{144 \text{ in.}^2}$$

$$p_{i_1} = 52.3 \text{ lbf/in.}^2$$

Similar calculations for Branches ② and ③ give

$$p_{i_2} = 66.3 \text{ lbf/in.}^2 \quad \text{and} \quad p_{i_3} = 43.3 \text{ lbf/in.}^2 \longleftarrow p_i$$

{ This problem illustrates the general method used to solve multiple-path pipe flow }
 { problems. }

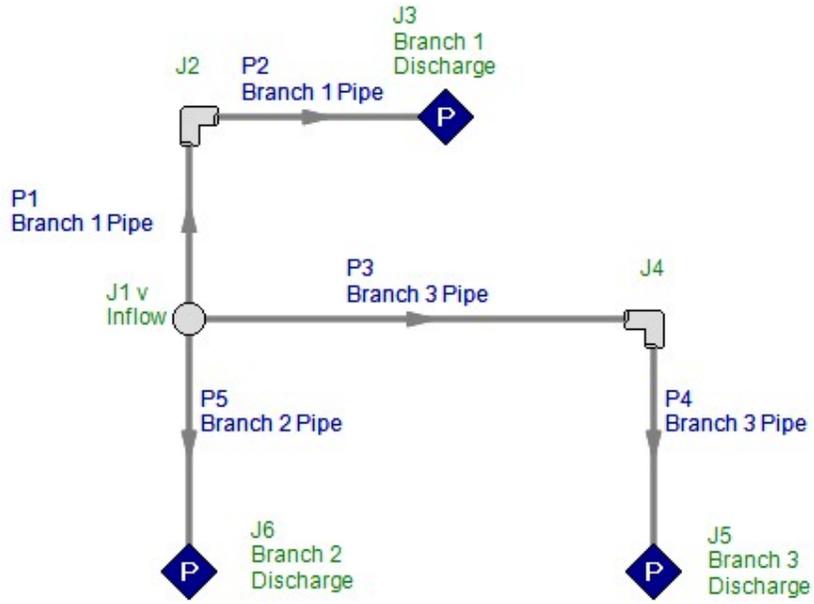
****8-8.3 Noncircular Ducts**

The empirical correlations for pipe flow also may be used for computations involving noncircular ducts, provided their cross sections are not too exaggerated. Thus ducts of square or rectangular cross section may be treated if the ratio of height to width is less than about 3 or 4.

** This section may be omitted without loss of continuity in the text material.

View Verification Case 41 Model

[Verification Case 41](#)



Verification Case 42

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify42.fth

REFERENCE: Theodore Baumeister, Eugene Avallone, Theodore Baumeister III, Marks' Standard Handbook for Mechanical Engineers, 8th ed., 1978, McGraw-Hill, Page 3-58

FLUID: Water at 68 deg. F

ASSUMPTIONS: Area change elevation is 130 feet, tanks are 10 feet deep

RESULTS:

Parameter	Marks'	AFT Fathom
Flow rate (ft ³ /sec)	0.4143	0.4144

DISCUSSION:

The fitting K factors were included in the pipes as a fitting and loss value.

[List of All Verification Models](#)

Verification Case 42 Problem Statement

Verification Case 42

Theodore Baumeister, Eugene Avallone, Theodore Baumeister III, Marks' Standard Handbook for Mechanical Engineers, 8th ed., 1978, McGraw-Hill, Page 3-58

Marks' Handbook Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

3-58 MECHANICS OF FLUIDS

Table 11. Representative Equivalent Length in Pipe Diameters (L/D) of Various Valves and Fittings

Globe valves, fully open	450
Angle valves, fully open	200
Gate valves, fully open	13
$\frac{3}{4}$ open	35
$\frac{1}{2}$ open	160
$\frac{1}{4}$ open	900
Swing check valves, fully open	135
In line, ball check valves, fully open	150
Butterfly valves, 6 in and larger, fully open	20
90° standard elbow	30
45° standard elbow	16
90° long-radius elbow	20
90° street elbow	50
45° street elbow	26
Standard tee:	
Flow through run	20
Flow through branch	60

Compiled from data given in "Flow of Fluids," Crane Company Technical Paper 410, ASME, 1971.

sizes, the practice is to group all of one size together and apply the continuity equation, as shown in the following example.

EXAMPLE. Water at 68°F (20°C) leaves an open tank whose surface elevation is 180 ft and enters a 2-in schedule 40 steel pipe via a sharp-edged entrance. After 50 ft of straight 2-in pipe that contains a 2-in globe valve, the line enlarges suddenly to an 8-in schedule 40 steel pipe which consists of 100 ft of straight 8-in pipe, two standard 90° elbows, and one 8-in-angle valve. The 8-in line discharges below the surface of another open tank whose surface elevation is 100 ft. Determine the volumetric flow rate.

$$D_1 = 2.067/12 = 0.1723 \quad \text{and} \quad D_2 = 7.981/12 = 0.6651$$

$$e/D_1 = 150 \times 10^{-6}/0.1723 = 8.706 \times 10^{-4}$$

$$e/D_2 = 150 \times 10^{-6}/0.6651 = 2.255 \times 10^{-4}$$

For turbulent flow,

$$\frac{1}{\sqrt{f_1}} = -2 \log_{10} \left(\frac{e/D_1}{3.7} \right)$$

$$\frac{1}{\sqrt{f_1}} = -2 \log_{10} \left(\frac{8.706 \times 10^{-4}}{3.7} \right) \quad f_1 = 0.01899$$

$$\frac{1}{\sqrt{f_2}} = -2 \log_{10} \left(\frac{2.255 \times 10^{-4}}{3.7} \right) \quad f_2 = 0.01407$$

1. 2-in components **K**

Entrance loss, sharp-edged entrance = 0.5

50 ft straight pipe = $f_1 (50/0.1723)$ = 290.2 f_1

Globe valve = $f_1 (L/D)$ = 450.0 f_1

Sudden enlargement $K = [1 - (D_1/D_2)^2]^2$ = 0.87

$\Sigma K_1 = 1.37 + 740.2 f_1$

2. 8-in components

100 ft of straight pipe $f_2 (100/0.6651)$ = 150.4 f_2

2 standard 90° elbows $2 \times 30 f_2$ = 60 f_2

1 angle valve $200 f_2$ = 200 f_2

Exit loss = 1

$\Sigma K_2 = 1 + 410.4 f_2$

3. Apply equation of motion

$$h_{1/2} = z_1 - z_2 = (\Sigma K_1) \frac{V_1^2}{2g} + (\Sigma K_2) \frac{V_2^2}{2g}$$

From continuity, $\rho_1 A_1 V_1 = \rho_2 A_2 V_2$ for $\rho_1 = \rho_2$

$$V_2 = V_1 (A_1/A_2) = V_1 (D_1/D_2)^2$$

$$h_{1/2} = z_1 - z_2 = [\Sigma K_1 + \Sigma K_2 (D_1/D_2)^4] V_1^2 / 2g$$

$$V_1 = \{ [2g(z_1 - z_2)] / (\Sigma K_1 + \Sigma K_2 (D_1/D_2)^4) \}^{1/2}$$

$$V_1 = \left[\frac{2 \times 32.17 \times (180 - 100)}{(1.37 + 740.2 f_1) + (1 + 410.4 f_2)(2.067/7.981)^4} \right]^{1/2}$$

$$V_1 = \frac{71.74}{(1.374 + 740.2 f_1 + 1.846 f_2)^{1/2}}$$

4. For first trial assume f_1 and f_2 for complete turbulence

$$V_1 = \frac{71.74}{(1.374 + 740.2 \times 0.01899 + 1.846 \times 0.01407)^{1/2}}$$

$$V_1 = 18.25 \text{ ft/s}$$

$$V_2 = 18.25 (2.067/7.981)^2 = 1.224 \text{ ft/s}$$

$$R_1 = \rho_1 V_1 D_1 / \mu = (1.937)(18.25)(0.1723) / (20.92 \times 10^{-6})$$

$$R_1 = 291,100 > 4,000 \therefore \text{flow is turbulent}$$

$$R_2 = \rho_2 V_2 D_2 / \mu_2 = (1.937)(1.224)(0.6651) / (20.92 \times 10^{-6}) \quad 1 \text{ pt.}$$

$$R_2 = 75,420 > 4,000 \therefore \text{flow is turbulent}$$

5. For second trial use first trial V_1 and V_2 . From Fig. 24 and the Colebrook equation,

$$\frac{1}{\sqrt{f_1}} = -2 \log_{10} \left(\frac{8.706 \times 10^{-4}}{3.7} + \frac{2.51}{291,100 \sqrt{0.020}} \right)$$

$$f_1 = 0.02008$$

$$\frac{1}{\sqrt{f_2}} = -2 \log_{10} \left(\frac{2.255 \times 10^{-4}}{3.7} + \frac{2.51}{75,420 \sqrt{0.020}} \right)$$

$$f_2 = 0.02008$$

$$V_1 = \frac{71.74}{(1.374 + 740.2 \times 0.02008 + 1.864 \times 0.02008)^{1/2}}$$

$$V_1 = 17.78$$

A third trial results in $V = 17.77$ ft/s or $Q = A_1 V_1 = (\pi/4)(0.1723)^2(17.77) = 0.4143$ ft³/s (1.173×10^{-2} m³/s).

Parallel Systems In solution of problems involving two or more parallel pipes, the head loss for each of the pipes is the same as shown in the following example:

EXAMPLE. Benzene at 68°F (20°C) flows at a rate of 0.5 ft³/s through two parallel straight, horizontal pipes connecting two pressurized tanks. The pipes are both schedule 40 steel, one being 1 in, the other 2 in. They both are 100 ft long and have connections that project inwardly in the supply tank. If the pressure in the supply tank is maintained at 100 lbf/in², what pressure should be maintained on the receiving tank?

$$D_1 = 1.049/12 = 0.08742 \text{ ft} \quad \text{and} \quad D_2 = 2.067/12 = 0.1723 \text{ ft}$$

$$e/D_1 = 150 \times 10^{-6}/0.08742 = 1.716 \times 10^{-3}$$

$$e/D_2 = 150 \times 10^{-6}/0.1723 = 8.706 \times 10^{-4}$$

For turbulent flow,

$$\frac{1}{\sqrt{f_1}} = -2 \log_{10} \left(\frac{e/D_1}{3.7} \right)$$

$$\frac{1}{\sqrt{f_1}} = -2 \log_{10} \left(\frac{1.716 \times 10^{-3}}{3.7} \right) \quad f_1 = 0.02249$$

$$\frac{1}{\sqrt{f_2}} = -2 \log_{10} \left(\frac{8.706 \times 10^{-4}}{3.7} \right) \quad f_2 = 0.01899$$

1. 1-in components **K**

Entrance loss, inward projection = 1.0

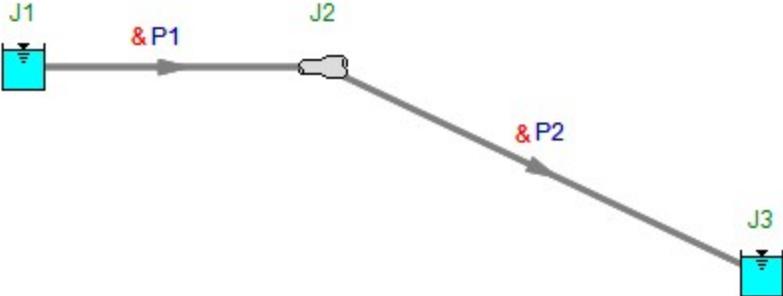
100 ft straight pipe $f_1 (100/0.08742)$ = 1,144 f_1

Exit loss = 1.0

$\Sigma K_1 = 2.0 + 1,144 f_1$

View Verification Case 42 Model

[Verification Case 42](#)



Verification Case 43

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify43.fth

REFERENCE: Theodore Baumeister, Eugene Avallone, Theodore Baumeister III, Marks' Standard Handbook for Mechanical Engineers, 8th ed., 1978, McGraw-Hill, Page 3-58

FLUID: Benzene at 68 deg. F

ASSUMPTIONS: N/A

RESULTS:

Parameter	Marks'	AFT Fathom
Flow rate in 1 inch pipe (ft ³ /sec)	0.0750	0.0744
Flow rate in 2 inch pipe (ft ³ /sec)	0.4250	0.4256
Pressure at discharge (psig)	73.40	73.28

DISCUSSION:

In order to fix the flowrate of 0.5 ft³/sec given in the problem statement the second pressurized tank is defined as a branch junction with a flow sink. In AFT Fathom the convention for flow out of the system is to use a negative sign, so the flow sink is entered as -0.5 ft³/sec.

[List of All Verification Models](#)

Verification Case 43 Problem Statement

Verification Case 43

Theodore Baumeister, Eugene Avallone, Theodore Baumeister III, Marks' Standard Handbook for Mechanical Engineers, 8th ed., 1978, McGraw-Hill, Page 3-58

Marks' Handbook Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

3-58 MECHANICS OF FLUIDS

Table 11. Representative Equivalent Length in Pipe Diameters (L/D) of Various Valves and Fittings

Globe valves, fully open	450
Angle valves, fully open	200
Gate valves, fully open	13
3/4 open	35
1/2 open	160
1/4 open	900
Swing check valves, fully open	135
In line, ball check valves, fully open	150
Butterfly valves, 6 in and larger, fully open	20
90° standard elbow	30
45° standard elbow	16
90° long-radius elbow	20
90° street elbow	20
45° street elbow	26
Standard tee:	
Flow through run	20
Flow through branch	60

Compiled from data given in "Flow of Fluids," Crane Company Technical Paper 410, ASME, 1971.

sizes, the practice is to group all of one size together and apply the continuity equation, as shown in the following example.

EXAMPLE. Water at 68°F (20°C) leaves an open tank whose surface elevation is 180 ft and enters a 2-in schedule 40 steel pipe via a sharp-edged entrance. After 50 ft of straight 2-in pipe that contains a 2-in globe valve, the line enlarges suddenly to an 8-in schedule 40 steel pipe which consists of 100 ft of straight 8-in pipe, two standard 90° elbows, and one 8-in-angle valve. The 8-in line discharges below the surface of another open tank whose surface elevation is 100 ft. Determine the volumetric flow rate.

$$D_1 = 2.067/12 = 0.1723 \quad \text{and} \quad D_2 = 7.981/12 = 0.6651$$

$$e/D_1 = 150 \times 10^{-6}/0.1723 = 8.706 \times 10^{-4}$$

$$e/D_2 = 150 \times 10^{-6}/0.6651 = 2.255 \times 10^{-4}$$

For turbulent flow,

$$\frac{1}{\sqrt{f_1}} = -2 \log_{10} \left(\frac{e/D}{3.7} \right)$$

$$\frac{1}{\sqrt{f_1}} = -2 \log_{10} \left(\frac{8.706 \times 10^{-4}}{3.7} \right) \quad f_1 = 0.01899$$

$$\frac{1}{\sqrt{f_2}} = -2 \log_{10} \left(\frac{2.255 \times 10^{-4}}{3.7} \right) \quad f_2 = 0.01407$$

1. 2-in components **K**

Entrance loss, sharp-edged entrance = 0.5

50 ft straight pipe f_1 (50/0.1723) = 290.2 f_1

Globe valve $= f_1 (L/D)$ = 450.0 f_1

Sudden enlargement $K = [1 - (D_1/D_2)^2]^2$ = 0.87

$= [1 - (2.067/7.981)^2]^2$ $\Sigma K_1 = 1.37 + 740.2 f_1$

2. 8-in components

100 ft of straight pipe f_2 (100/0.6651) = 150.4 f_2

2 standard 90° elbows $2 \times 30 f_2$ = 60 f_2

1 angle valve $200 f_2$ = 200 f_2

Exit loss = 1

$\Sigma K_2 = 1 + 410.4 f_2$

3. Apply equation of motion

$$h_{1/2} = z_1 - z_2 = (\Sigma K_1) \frac{V_1^2}{2g} + (\Sigma K_2) \frac{V_2^2}{2g}$$

From continuity, $\rho_1 A_1 V_1 = \rho_2 A_2 V_2$ for $\rho_1 = \rho_2$

$$V_2 = V_1 (A_1/A_2) = V_1 (D_1/D_2)^2$$

$$h_{1/2} = z_1 - z_2 = (\Sigma K_1 + \Sigma K_2 (D_1/D_2)^4) V_1^2 / 2g$$

$$V_1 = \{ [2g(z_1 - z_2)] / (\Sigma K_1 + \Sigma K_2 (D_1/D_2)^4) \}^{1/2}$$

$$V_1 = \left[\frac{2 \times 32.17 \times (180 - 100)}{(1.37 + 740.2 f_1) + (1 + 410.4 f_2)(2.067/7.981)^4} \right]^{1/2}$$

$$V_1 = \frac{71.74}{(1.374 + 740.2 f_1 + 1.846 f_2)^{1/2}}$$

4. For first trial assume f_1 and f_2 for complete turbulence

$$V_1 = \frac{71.74}{(1.374 + 740.2 \times 0.01899 + 1.846 \times 0.01407)^{1/2}}$$

$$V_1 = 18.25 \text{ ft/s}$$

$$V_2 = 18.25 (2.067/7.981)^2 = 1.224 \text{ ft/s}$$

$$R_1 = \rho_1 V_1 D_1 / \mu = (1.937)(18.25)(0.1723) / (20.92 \times 10^{-6})$$

$$R_1 = 291,100 > 4,000 \therefore \text{flow is turbulent}$$

$$R_2 = \rho_2 V_2 D_2 / \mu_2 = (1.937)(1.224)(0.6651) / (20.92 \times 10^{-6}) \quad 1 \text{ pt.}$$

$$R_2 = 75,420 > 4,000 \therefore \text{flow is turbulent}$$

5. For second trial use first trial V_1 and V_2 . From Fig. 24 and the Colebrook equation,

$$\frac{1}{\sqrt{f_1}} = -2 \log_{10} \left(\frac{8.706 \times 10^{-4}}{3.7} + \frac{2.51}{291,100 \sqrt{0.020}} \right)$$

$$f_1 = 0.02008$$

$$\frac{1}{\sqrt{f_2}} = -2 \log_{10} \left(\frac{2.255 \times 10^{-4}}{3.7} + \frac{2.51}{75,420 \sqrt{0.020}} \right)$$

$$f_2 = 0.02008$$

$$V_1 = \frac{71.74}{(1.374 + 740.2 \times 0.02008 + 1.864 \times 0.02008)^{1/2}}$$

$$V_1 = 17.78$$

A third trial results in $V = 17.77$ ft/s or $Q = A_1 V_1 = (\pi/4)(0.1723)^2(17.77) = 0.4143$ ft³/s (1.173×10^{-2} m³/s).

Parallel Systems In solution of problems involving two or more parallel pipes, the head loss for each of the pipes is the same as shown in the following example:

EXAMPLE. Benzene at 68°F (20°C) flows at a rate of 0.5 ft³/s through two parallel straight, horizontal pipes connecting two pressurized tanks. The pipes are both schedule 40 steel, one being 1 in, the other 2 in. They both are 100 ft long and have connections that project inwardly in the supply tank. If the pressure in the supply tank is maintained at 100 lbf/in², what pressure should be maintained on the receiving tank?

$$D_1 = 1.049/12 = 0.08742 \text{ ft} \quad \text{and} \quad D_2 = 2.067/12 = 0.1723 \text{ ft}$$

$$e/D_1 = 150 \times 10^{-6}/0.08742 = 1.716 \times 10^{-3}$$

$$e/D_2 = 150 \times 10^{-6}/0.1723 = 8.706 \times 10^{-4}$$

For turbulent flow,

$$\frac{1}{\sqrt{f_1}} = -2 \log_{10} \left(\frac{e/D}{3.7} \right)$$

$$\frac{1}{\sqrt{f_1}} = -2 \log_{10} \left(\frac{1.716 \times 10^{-3}}{3.7} \right) \quad f_1 = 0.02249$$

$$\frac{1}{\sqrt{f_2}} = -2 \log_{10} \left(\frac{8.706 \times 10^{-4}}{3.7} \right) \quad f_2 = 0.01899$$

1. 1-in components **K**

Entrance loss, inward projection = 1.0

100 ft straight pipe f_1 (100/0.08742) = 1,144 f_1

Exit loss = 1.0

$\Sigma K_1 = 2.0 + 1,144 f_1$

Verification Case 43 Problem Statement

PIPING SYSTEMS 3-59

2. 2-in components
- Entrance loss, inward projection = 1.0
 - 100 ft straight pipe f_2 (100/0.1723) = 580.4 f_2
 - Exit loss = 1.0
- $\Sigma K_2 = 2.0 + 580.4 f_2$

$$b_f = \Sigma K_1 V^2 / 2g = \Sigma K_2 V^2 / 2g$$

From the continuity equation, $Q = AV$

$$\Sigma K_1 \frac{Q^2}{2gA_1^5} = \Sigma K_2 \frac{Q^2}{2gA_2^5}$$

Solving for Q_1/Q_2 ,

$$\frac{Q_1}{Q_2} = \frac{A_2}{A_1} \sqrt{\frac{\Sigma K_2}{\Sigma K_1}} = \left(\frac{D_1}{D_2}\right)^2 \sqrt{\frac{\Sigma K_2}{\Sigma K_1}} = \left(\frac{D_1}{D_2}\right)^2 \sqrt{\frac{2.0 + 580.4 f_2}{2.0 + 1,144 f_1}}$$

For first trial assume flow is completely turbulent,

$$\frac{Q_1}{Q_2} = \left(\frac{0.08742}{0.1723}\right)^2 \sqrt{\frac{2.0 + 580.4 \times 0.01899}{2.0 + 1,144 \times 0.02249}}$$

$$\frac{Q_1}{Q_2} = 0.1764 \quad Q = Q_1 + Q_2 = 0.1764 Q_2 + Q_2$$

$$0.5000 = 1.1764 Q_2 \quad Q_2 = 0.4250$$

$$Q_1 = 0.5000 - 0.4250 = 0.0750$$

For second trial use first-trial values,

$$V_1 = Q_1/A_1 = 0.0750/(\pi/4)(0.08742)^2 = 12.50$$

$$V_2 = Q_2/A_2 = 0.4250/(\pi/4)(0.1723)^2 = 18.23$$

$$R_1 = \rho_1 V_1 D_1 / \mu_1 = (1.705)(12.50)(0.08742)/(13.62 \times 10^{-6})$$

$$R_1 = 136,800 > 4,000 \therefore \text{flow is turbulent}$$

$$R_2 = \rho_2 V_2 D_2 / \mu_2 = (1.705)(18.23)(0.1723)/(13.62 \times 10^{-6})$$

$$R_2 = 393,200 > 4,000 \therefore \text{flow is turbulent}$$

Using the Colebrook equation and Fig. 24,

$$\frac{1}{\sqrt{f_1}} = -2 \log_{10} \left(\frac{1.716 \times 10^{-2}}{3.7} + \frac{2.51}{136,800 \sqrt{0.024}} \right)$$

$$f_1 = 0.02389$$

$$\frac{1}{\sqrt{f_2}} = -2 \log_{10} \left(\frac{8.706 \times 10^{-4}}{3.7} + \frac{2.51}{393,200 \sqrt{0.020}} \right)$$

$$f_2 = 0.01981$$

$$b_f = \Sigma K_1 \frac{V_1^2}{2g} = \Sigma K_2 \frac{V_2^2}{2g}$$

$$= \frac{\Sigma K_1 V_1^2 / 2g}{\Sigma K_2 V_2^2 / 2g} = (2.0 + 1,144 \times 0.02389)(12.50)^2 / (2 \times 32.17) = 71.23$$

$$= \frac{\Sigma K_2 V_2^2 / 2g}{\Sigma K_2 V_2^2 / 2g} = (2.0 + 580.4 \times 0.01981)(18.23)^2 / (2 \times 32.17) = 69.80$$

71.23 = 69.80; further trials not justifiable because of accuracy of f , K , L/D . Use average or 70.52, so that $\Delta p = \rho g b_f = (1.705 \times 32.17 \times 70.52)(144) = 26.86 \text{ lbf/in}^2 = p_1 - p_2 = 100 - p_2$, $p_2 = 100 - 26.86 = 73.14 \text{ lbf/in}^2$ ($5.061 \times 10^6 \text{ N/m}^2$).

Branch Flow Problems of a single line feeding several points may be solved as shown in the following example.

EXAMPLE. Ethyl alcohol at 68°F (20°C) flows from tank A, which is maintained at a constant pressure of 100 lb/in² through 200 ft of 2-in cast-iron schedule 40 pipe to a Y branch connection ($K = 0.5$) where 100 ft of 2-in pipe goes to tank B, which is maintained at 80 lb/in² and 50 ft of 2-in pipe goes to tank C, which is also maintained at 80 lb/in². All tank connections are flush and sharp-edged and are at the same elevation. Estimate the flow rate to each tank.

$$D = 2.067/12 = 0.1723 \text{ ft}$$

$$e/D = 850 \times 10^{-6}/0.1723 = 4.933 \times 10^{-3}$$

For turbulent flow, $\frac{1}{\sqrt{f}} = -2 \log_{10} \left(\frac{e/D}{3.7} \right)$

$$\frac{1}{\sqrt{f}} = -2 \log_{10} \left(\frac{4.933 \times 10^{-3}}{3.7} \right) \quad f = 0.03025$$

$$b_{A/B} = (\rho_A - \rho_B) \rho g = 144(100 - 80)(1.532 \times 32.17) = 58.44$$

$$b_{A/C} = (\rho_A - \rho_C) \rho g = b_{A/B} = 58.44$$

Let point X be just before the Y; then

1. From tank A to Y
 - Entrance loss, sharp-edged = 0.5
 - 200 ft straight pipe = $f_{AX}(200/0.1723)$ = 1,161 f_{AX}

$\Sigma K_{AX} = 0.5 + 1,161 f_{AX}$
2. From Y to tank B
 - Y branch = 0.5
 - 100 ft straight pipe = $f_{XB}(100/0.1723)$ = 580.4 f_{XB}
 - Exit loss = 1.0

$\Sigma K_{XB} = 1.5 + 580.4 f_{XB}$
3. From Y to tank C
 - Y branch = 0.5
 - 50 ft straight pipe = $f_{XC}(50/0.1723)$ = 290.2 f_{XC}
 - Exit loss = 1.0

$\Sigma K_{XC} = 1.5 + 290.2 f_{XC}$

Balance of flows:

$$Q_{AX} = Q_{XB} + Q_{XC}$$

and from continuity, ($A_{AX} = A_{XB} = A_{XC}$), $V_{AX} = V_{XB} + V_{XC}$; then

$$b_{A/B} = \Sigma K_{AX} \frac{V_{AX}^2}{2g} + \Sigma K_{XB} \frac{V_{XB}^2}{2g}$$

$$b_{A/C} = \Sigma K_{AX} \frac{V_{AX}^2}{2g} + \Sigma K_{XC} \frac{V_{XC}^2}{2g}$$

For first trial assume completely turbulent flow

$$b_{A/B} = \frac{(0.5 + 1,161 f_{AX}) V_{AX}^2}{2g} + \frac{(1.5 + 580.4 f_{XB}) V_{XB}^2}{2g}$$

$$58.44 = \frac{(0.5 + 1,161 \times 0.03025) V_{AX}^2}{2 \times 32.17} + \frac{(1.5 + 580.4 \times 0.03025) V_{XB}^2}{2 \times 32.17}$$

$$58.44 = 0.5536 V_{AX}^2 + 0.2962 V_{XB}^2$$

and in a like manner

$$b_{A/C} = 58.44 = 0.5536 V_{AX}^2 + 0.1598 V_{XC}^2$$

Equating $b_{A/B} = b_{A/C}$,

$$0.5536 V_{AX}^2 + 0.2962 V_{XB}^2 = 0.5536 V_{AX}^2 + 0.1598 V_{XC}^2$$

or $V_{XC} = 1.3615 V_{XB}$ and since $V_{AX} = V_{XB} + V_{XC}$

$$V_{AX} = V_{XB} + 1.3615 V_{XB} = 2.3615 V_{XB}$$

so that

$$b_{A/B} = 58.44 = 0.5536(2.3615 V_{XB})^2 + 0.2962 V_{XB}^2$$

$$V_{XB} = 4.156$$

$$V_{XC} = 1.3615(4.156) = 5.658$$

$$V_{AX} = 4.156 + 5.658 = 9.814$$

Second trial,

$$R_{AX} = \frac{\rho V_{AX} D}{\mu} = \frac{1.532 \times 9.814 \times 0.1723}{25.06 \times 10^{-6}}$$

$$R_{AX} = 103,400 > 4,000 \therefore \text{flow is turbulent}$$

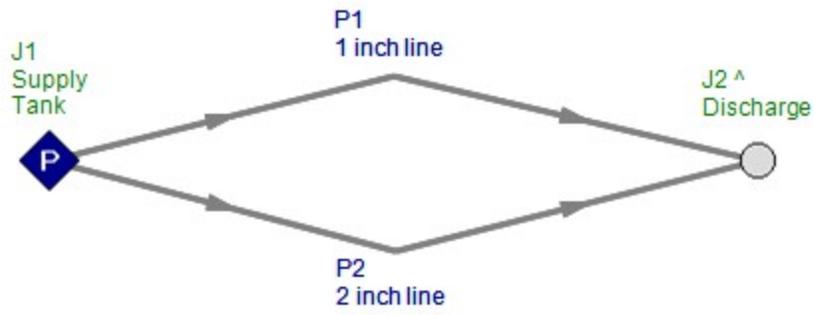
In a like manner,

$$R_{XB} = 43,780 \quad R_{XC} = 59,600$$

Using the Colebrook equation and Fig. 24,

View Verification Case 43 Model

[Verification Case 43](#)



Verification Case 44

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify44.fth

REFERENCE: Theodore Baumeister, Eugene Avallone, Theodore Baumeister III, Marks' Standard Handbook for Mechanical Engineers, 8th ed., 1978, McGraw-Hill, Page 3-59, 60

FLUID: Ethyl Alcohol (Ethanol) at 68 deg. F

ASSUMPTIONS: N/A

RESULTS:

Parameter	Marks'	AFT Fathom
Flow rate to Tank B (ft ³ /sec)	0.09491	0.09474
Flow rate to Tank C (ft ³ /sec)	0.1304	0.13023

DISCUSSION:

All inputs were provided explicitly by the problem statement.

[List of All Verification Models](#)

Verification Case 44 Problem Statement

Verification Case 44

Theodore Baumeister, Eugene Avallone, Theodore Baumeister III, Marks' Standard Handbook for Mechanical Engineers, 8th ed., 1978, McGraw-Hill, Page 3-59, 60

Marks' Handbook Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

PIPING SYSTEMS 3-59

2. 2-in components
 Entrance loss, inward projection = 1.0
 100 ft straight pipe f_2 (100/0.1723) = 580.4 f_2
 Exit loss = 1.0
 $\Sigma K_2 = 2.0 + 580.4 f_2$

$b_f = \Sigma K_1 V_1^2 / 2g = \Sigma K_2 V_2^2 / 2g$
 From the continuity equation, $Q = AV$
 $\Sigma K_1 \frac{Q^2}{2gA_1^5} = \Sigma K_2 \frac{Q^2}{2gA_2^5}$

Solving for Q_1/Q_2 ,

$$\frac{Q_1}{Q_2} = \frac{A_2}{A_1} \sqrt{\frac{\Sigma K_2}{\Sigma K_1}} = \left(\frac{D_2}{D_1}\right)^2 \sqrt{\frac{\Sigma K_2}{\Sigma K_1}} = \left(\frac{D_1}{D_2}\right)^2 \sqrt{\frac{2.0 + 580.4 f_2}{2.0 + 1,144 f_1}}$$

For first trial assume flow is completely turbulent,

$$\frac{Q_1}{Q_2} = \frac{(0.08742)^2}{(0.1723)^2} \sqrt{\frac{2.0 + 580.4 \times 0.01899}{2.0 + 1,144 \times 0.02249}}$$

$$\frac{Q_1}{Q_2} = 0.1764 \quad Q = Q_1 + Q_2 = 0.1764 Q_2 + Q_2$$

0.5000 = 1.1764 Q_2 $Q_2 = 0.4250$
 $Q_1 = 0.5000 - 0.4250 = 0.0750$

For second trial use first-trial values,

$V_1 = Q_1/A_1 = 0.0750/(\pi/4)(0.08742)^2 = 12.50$
 $V_2 = Q_2/A_2 = 0.4250/(\pi/4)(0.1723)^2 = 18.23$
 $R_1 = \rho_1 V_1 D_1 / \mu_1 = (1.705)(12.50)(0.08742)/(13.62 \times 10^{-6})$
 $R_1 = 136,800 > 4,000 \therefore$ flow is turbulent
 $R_2 = \rho_2 V_2 D_2 / \mu_2 = (1.705)(18.23)(0.1723)/(13.62 \times 10^{-6})$
 $R_2 = 393,200 > 4,000 \therefore$ flow is turbulent

Using the Colebrook equation and Fig. 24,

$$\frac{1}{\sqrt{f_1}} = -2 \log_{10} \left(\frac{1.716 \times 10^{-2}}{3.7} + \frac{2.51}{136,800 \sqrt{0.024}} \right)$$

$$f_1 = 0.02389$$

$$\frac{1}{\sqrt{f_2}} = -2 \log_{10} \left(\frac{8.706 \times 10^{-4}}{3.7} + \frac{2.51}{393,200 \sqrt{0.020}} \right)$$

$$f_2 = 0.01981$$

$$b_f = \Sigma K_1 \frac{V_1^2}{2g} = \Sigma K_2 \frac{V_2^2}{2g}$$

$$\Sigma K_1 V_1^2 / 2g = \Sigma K_2 V_2^2 / 2g$$

$$= (2.0 + 1,144 \times 0.02389)(12.50)^2 / (2 \times 32.17) = 71.23$$

$$\Sigma K_2 V_2^2 / 2g = (2.0 + 580.4 \times 0.01981)(18.23)^2 / (2 \times 32.17) = 69.80$$

71.23 = 69.80; further trials not justifiable because of accuracy of f , K , L/D . Use average or 70.52, so that $\Delta p = \rho g b_f = (1.705 \times 32.17 \times 70.52) / 144 = 26.86$ lbf/in² = $p_1 - p_2 = 100 - p_2$, $p_2 = 100 - 26.86 = 73.14$ lbf/in² (5.061×10^5 N/m²).

Branch Flow Problems of a single line feeding several points may be solved as shown in the following example.

EXAMPLE. Ethyl alcohol at 68°F (20°C) flows from tank A, which is maintained at a constant pressure of 100 lbf/in² through 200 ft of 2-in cast-iron schedule 40 pipe to a Y branch connection ($K = 0.5$) where 100 ft of 2-in pipe goes to tank B, which is maintained at 80 lbf/in² and 50 ft of 2-in pipe to tank C, which is also maintained at 80 lbf/in². All tank connections are flush and sharp-edged and are at the same elevation. Estimate the flow rate to each tank.

$D = 2.067/12 = 0.1723$ ft
 $\epsilon/D = 850 \times 10^{-6}/0.1723 = 4.933 \times 10^{-3}$

For turbulent flow, $\frac{1}{\sqrt{f}} = -2 \log_{10} \left(\frac{\epsilon/D}{3.7} \right)$
 $\frac{1}{\sqrt{f}} = -2 \log_{10} \left(\frac{4.933 \times 10^{-3}}{3.7} \right) \quad f = 0.03025$
 $b_{A/B} = (\rho_A - \rho_B) \rho g = 144(100 - 80)/(1.532 \times 32.17) = 58.44$
 $b_{A/C} = (\rho_A - \rho_C) \rho g = b_{A/B} = 58.44$

Let point X be just before the Y; then

- From tank A to Y
 Entrance loss, sharp-edged = 0.5
 200 ft straight pipe = $f_{AX}(200/0.1723)$ = 1,161 f_{AX}
 $\Sigma K_{AX} = 0.5 + 1,161 f_{AX}$
- From Y to tank B
 Y branch = 0.5
 100 ft straight pipe = $f_{XB}(100/0.1723)$ = 580.4 f_{XB}
 Exit loss = 1.0
 $\Sigma K_{XB} = 1.5 + 580.4 f_{XB}$
- From Y to tank C
 Y branch = 0.5
 50 ft straight pipe = $f_{XC}(50/0.1723)$ = 290.2 f_{XC}
 Exit loss = 1.0
 $\Sigma K_{XC} = 1.5 + 290.2 f_{XC}$

Balance of flows:

$$Q_{AX} = Q_{XB} + Q_{XC}$$

and from continuity, ($A_{AX} = A_{XB} = A_{XC}$), $V_{AX} = V_{XB} + V_{XC}$; then

$$b_{A/B} = \Sigma K_{AX} \frac{V_{AX}^2}{2g} + \Sigma K_{XB} \frac{V_{XB}^2}{2g}$$

$$b_{A/C} = \Sigma K_{AX} \frac{V_{AX}^2}{2g} + \Sigma K_{XC} \frac{V_{XC}^2}{2g}$$

For first trial assume completely turbulent flow

$$b_{A/B} = (0.5 + 1,161 f_{AX}) V_{AX}^2 + (1.5 + 580.4 f_{XB}) V_{XB}^2$$

$$58.44 = \frac{(0.5 + 1,161 \times 0.03025) V_{AX}^2}{2 \times 32.17} + \frac{(1.5 + 580.4 \times 0.03025) V_{XB}^2}{2 \times 32.17}$$

$$58.44 = 0.5536 V_{AX}^2 + 0.2962 V_{XB}^2$$

and in a like manner

$$b_{A/C} = 58.44 = 0.5536 V_{AX}^2 + 0.1598 V_{XC}^2$$

Equating $b_{A/B} = b_{A/C}$

$$0.5536 V_{AX}^2 + 0.2962 V_{XB}^2 = 0.5536 V_{AX}^2 + 0.1598 V_{XC}^2$$

or $V_{XC} = 1.3615 V_{XB}$ and since $V_{AX} = V_{XB} + V_{XC}$

$$V_{AX} = V_{XB} + 1.3615 V_{XB} = 2.3615 V_{XB}$$

so that

$$b_{A/B} = 58.44 = 0.5536(2.3615 V_{XB})^2 + 0.2962 V_{XB}^2$$

$$V_{XB} = 4.156$$

$$V_{XC} = 1.3615(4.156) = 5.658$$

$$V_{AX} = 4.156 + 5.658 = 9.814$$

Second trial,

$$R_{AX} = \frac{\rho V_{AX} D}{\mu} = \frac{1.532 \times 9.814 \times 0.1723}{25.06 \times 10^{-6}}$$

$$R_{AX} = 103,400 > 4,000 \therefore$$
 flow is turbulent

In a like manner,

$$R_{XB} = 43,780 \quad R_{XC} = 59,600$$

Using the Colebrook equation and Fig. 24,

Verification Case 44 Problem Statement

3-60 MECHANICS OF FLUIDS

$$\frac{1}{f_{Ax}} = -2 \log_{10} \left(\frac{4.933 \times 10^{-3}}{3.7} + \frac{2.51}{103,400 \sqrt{0.031}} \right)$$

$$f_{Ax} = 0.03116$$

In a like manner,

$$f_{xB} = 0.03231 \quad f_{xC} = 0.03179$$

$$b_{A/B} = \frac{(0.5 + 1.161 \times 0.03116) V_{Ax}^2}{2 \times 32.17} + \frac{(1.5 + 580.4 \times 0.03231) V_{xB}^2}{2 \times 32.17}$$

$$b_{A/B} = 0.5700 V_{Ax}^2 + 0.3148 V_{xB}^2$$

$$b_{A/C} = 0.5700 V_{Ax}^2 + \frac{(1.5 + 290.2 \times 0.03179) V_{xC}^2}{2 \times 32.17}$$

$$b_{A/C} = 0.5700 V_{Ax}^2 + 0.1667 V_{xC}^2$$

$$0.3148 V_{xB}^2 = 0.1667 V_{xC}^2$$

$$V_{xC} = 1.374 V_{xB}$$

$$V_{Ax} = V_{xB} + 1.374 V_{xB} = 2.374 V_{xB}$$

so that

$$b_{A/B} = 58.44 = 0.5700 (2.374 V_{xB})^2 + 0.3148 V_{xB}^2$$

$$V_{xB} = 4.070 \quad V_{xC} = 5.592 \quad V_{Ax} = 9.663$$

Further trials are not justified.

$$A = \pi D^2/4 = (\pi/4)(0.1723)^2 = 0.02332 \text{ ft}^2$$

$$Q_{xB} = V_{xB} A = 4.070 \times 0.02332 = 0.09491 \text{ ft}^3/\text{s} \quad (2.686 \times 10^{-3} \text{ m}^3/\text{s})$$

$$Q_{xC} = V_{xC} A = 5.592 \times 0.02332 = 0.1304 \text{ ft}^3/\text{s} \quad (3.693 \times 10^{-3} \text{ m}^3/\text{s})$$

Siphons are arrangements of hose or pipe which cause liquids to flow from one level A in Fig. 25 to a lower level C over an intermediate summit B. Performance of siphons may be evaluated from the equation of motion between points A and B:

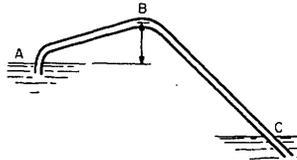


Fig. 25 Siphon.

ated from the equation of motion between points A and B:

$$\frac{p_A}{\gamma} + \frac{V_A^2}{2g} + z_A = \frac{p_B}{\gamma} + \frac{V_B^2}{2g} + z_B + b_{A/B}$$

Noting that on the surface $V_A = 0$ and the minimum pressure that can exist at point B is the vapor pressure p_v , the maximum elevation of point B is

$$z_B - z_A = \frac{p_A - p_v}{\gamma} - \left(\frac{V_B^2}{2g} + b_{A/B} \right)$$

The friction loss $b_f = \sum K_{AB} V^2/2g$, and let $V_B = V$; then

$$z_B - z_A = \frac{p_A - p_v}{\gamma} - (1 + \sum K_{AB}) \frac{V^2}{2g}$$

Flow under this maximum condition will be uncertain. The air pump or ejector used for priming the pipe (flow will not take place unless the siphon is full of water) might have to be operated occasionally to remove accumulated air and vapor. Values of $z_B - z_A$ less than those calculated by the above equation should be used.

EXAMPLE. The siphon shown in Fig. 25 is composed of 2,000 ft of

6-in schedule 40 cast-iron pipe. Reservoir A is at elevation 800 ft and C at 600 ft. Estimate the maximum height for $z_B - z_A$ if the water temperature may reach 104°F (40°C), and the amount of straight pipe from A to B is 100 ft. For the first bend $L/D = 25$ and the second (at B) $L/D = 50$. Atmospheric pressure is 14.70 lbf/in². For 6-in schedule 40 pipe $D = 6.065/12 = 0.5054$ ft, $e/D = 850 \times 10^{-6}/0.5054 = 1.682 \times 10^{-3}$. Turbulent friction factor $1/f = -2 \log_{10} \left(\frac{e/D}{3.7} \right) = -2 \log_{10} (1.682 \times 10^{-3}/3.7) = 0.02238$.

1. Components from A to B K
(Note loss in second bend takes place in downstream piping)
Entrance (inward projection) = 1.0
100 ft straight pipe $f(100/0.5054)$ = 197.9 f
First bend = 25 f
 $\Sigma K_{AB} = 1.0 + 227.9 f$
2. Components from A to C
 ΣK_{AC} = 1.0 + 2,229 f
1,900 ft of straight pipe $f(1,900/0.5054)$ = 3,759.4 f
Second bend = 50 f
Exit loss = 1
 $\Sigma K_{AC} = 2.0 + 4,032 f$

First trial assume complete turbulence. Writing the equation of motion between A and C,

$$\frac{p_A}{\gamma} + \frac{V_A^2}{2g} + z_A = \frac{p_C}{\gamma} + \frac{V_C^2}{2g} + z_C + \Sigma K_{AC} \frac{V^2}{2g}$$

Noting $V_A = V_C = 0$, and $p_A = p_C = 14.7$ lbf/in²,

$$V = \sqrt{\frac{2g(z_A - z_C)}{\Sigma K_{AC}}} = \sqrt{\frac{2g(800 - 600)}{2.0 + 4,032 f}} = \sqrt{\frac{2 \times 32.17(800 - 600)}{2.0 + 4,032 f}}$$

$$= \frac{113.44}{\sqrt{2.0 + 4,032 \times 0.02238}} = 11.81$$

Second trial, use first trial values,

$$R = \frac{\rho V D}{\mu} = (1.925)(11.81)(0.5054)/(13.61 \times 10^{-6})$$

$$R = 846,200 > 4,000 \therefore \text{flow is turbulent}$$

From Fig. 24 and the Colebrook equation,

$$\frac{1}{f} = -2 \log_{10} \left(\frac{1.682 \times 10^{-3}}{3.7} + \frac{2.51}{844,200 \sqrt{0.023}} \right)$$

$$f = 0.02263$$

$$V = \frac{113.44}{\sqrt{2.0 + 4,032 \times 0.02263}} = 11.75 \quad (\text{close check})$$

From Sec. 4 steam tables at 104°F, $p_v = 1.070$ lbf/in², the maximum height

$$z_B - z_A = \frac{p_A - p_v}{\gamma} - (1 + \Sigma K_{AB}) \frac{V^2}{2g}$$

$$z_B - z_A = \frac{144(14.70 - 1.070)}{1.925 \times 32.17} - (1 + 1 + 227.9 \times 0.02262) \frac{(11.75)^2}{2 \times 32.17} = 16.58 \text{ ft} \quad (5.053 \text{ m})$$

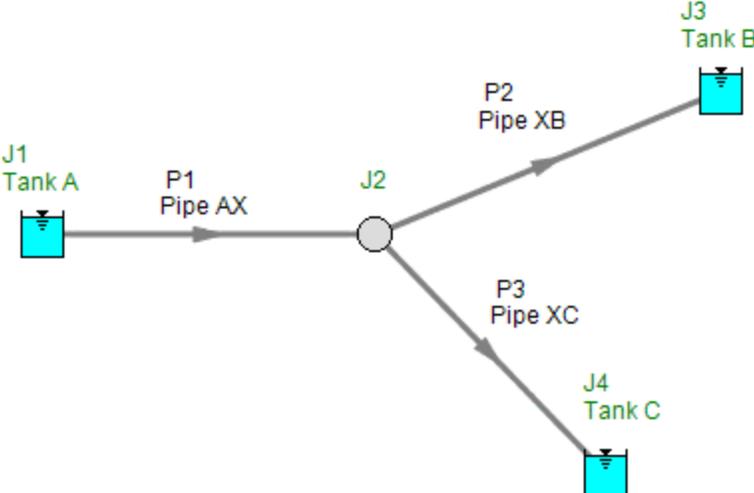
Note that if a ± 10 percent error exists in calculation of pressure loss, maximum height should be limited to ≈ 15 ft (5 m).

ASME PIPELINE FLOWMETERS

Parameters Dimensional analysis of the flow of an incompressible fluid flowing in a pipe of diameter D , surface rough-

View Verification Case 44 Model

[Verification Case 44](#)



Verification Case 45

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify45.fth

REFERENCE: James John, William Haberman, Introduction to Fluid Mechanics, 2nd Ed., 1980, Prentice-Hall, Page 168-169, Example 6.3

FLUID: Water at 25 deg. C

ASSUMPTIONS: Flow rate is 0.01 m³/sec (omitted in problem statement)

RESULTS:

Parameter	John & Haberman	AFT Fathom
Pressure drop in cast iron pipe (kPa)	21.81	21.90
Prssure drop in pipe 2 (kPa)	13.46	13.63

DISCUSSION:

Results disagree slightly because John & Haberman read friction factors from a chart, which is less accurate than AFT Fathom's correlation based method.

[List of All Verification Models](#)

Verification Case 45 Problem Statement

[Verification Case 45](#)

James John, William Haberman, Introduction to Fluid Mechanics, 2nd Ed., 1980, Prentice-Hall, Page 168-169, Example 6.3

[John and Haberman Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

168

Pipe Flow

Note that for large values of relative roughness, the dividing line between the two regions occurs at a lower Reynolds number than for small values of relative roughness. This is because the thickness of the laminar sublayer decreases as the Reynolds number, and hence the turbulence of the flow, increases. So, at low Reynolds numbers, with a relatively thick laminar sublayer, only very rough pipes will have protuberances that will project through the sublayer.

In order to determine a value of the friction factor f from the Moody diagram, a knowledge of relative roughness is necessary. Typical values of roughness for various types of pipe are shown in Table 6.1.

Table 6.1.

Type	ϵ (mm)
Glass	Smooth
Asphalted cast iron	0.12
Galvanized iron	0.15
Cast iron	0.26
Wood stave	0.18–0.90
Concrete	0.30–3.0
Riveted steel	1.0–10
Drawn tubing	0.0015

It should be noted that after pipes have been in service for a time, deposits build up on the pipe walls which may substantially increase the preceding values of ϵ .

For noncircular pipes, good correlation for fully developed turbulent pipe flow is obtained by using Figure 6.18, with Re_{D_h} substituted for Re_D and ϵ/D_h for ϵ/D . The values of critical Reynolds number from Section 6.1 can also be used, with Re_{D_h} in place of Re_D .

For laminar flow in noncircular pipes, however, an appreciable error can result from using the value of f for circular pipes, as was shown for flow through the rectangular channel worked out previously in this section. The magnitude of the error is dependent on the shape of the cross section.

The following example illustrates the use of the Moody diagram.

EXAMPLE 6.3. Ten liters per second of water at 25°C is to flow through a horizontal pipe 100 m long. Compare the pressure drop in a cast iron pipe of circular cross section of 10 cm diameter with the pressure drop in a wood pipe ($\epsilon = 0.30$ mm) of square cross section 10 cm on a side.

Solution: The water velocity in the circular pipe, \bar{V} , is given by

$$\bar{V} = \frac{10 \times 10^{-3} \text{ m}^3/\text{s}}{(\pi/4)(0.10)^2 \text{ m}^2} = 1.273 \text{ m/s}$$

At 25°C, $\nu = 0.898 \times 10^{-6} \text{ m}^2/\text{s}$, so that

$$\text{Re}_D = \frac{(1.273 \text{ m/s})(0.10) \text{ m}}{0.898 \times 10^{-6} \text{ m}^2/\text{s}} = 1.418 \times 10^5$$

This is well over the critical value of 2200 given in Section 6.1, so we are in the region of fully developed turbulent pipe flow. In order to find f from Figure 6.18, we need relative roughness:

$$\frac{\epsilon}{D} = \frac{0.30 \times 10^{-3} \text{ m}}{0.10 \text{ m}} = 0.0030 \quad \text{for the circular pipe}$$

With relative roughness and Reynolds number, we now go to the Moody diagram to find $f = 0.027$. From (6.5),

$$\begin{aligned} p_2 - p_1 &= -\frac{1}{2} \rho \bar{V}^2 \frac{fL}{D} \\ &= -\frac{1}{2} (997 \text{ kg/m}^3)(1.273)^2 \text{ m}^2/\text{s}^2 \left(\frac{0.027 \times 100 \text{ m}}{0.10 \text{ m}} \right) \\ &= \underline{-21.81 \text{ kPa}} \end{aligned}$$

For the square wood pipe,

$$\begin{aligned} \bar{V} &= \frac{10 \times 10^{-3} \text{ m}^3/\text{s}}{(0.10)^2 \text{ m}^2} = 1.0 \text{ m/s} \\ D_h &= 10 \text{ cm} \quad (\text{see Section 6.2}) \\ \text{Re}_{D_h} &= \frac{(1.0 \text{ m/s})(0.10) \text{ m}}{0.898 \times 10^{-6} \text{ m}^2/\text{s}} = 1.114 \times 10^5 \\ \frac{\epsilon}{D_h} &= \frac{0.30 \times 10^{-3} \text{ m}}{0.10 \text{ m}} = 0.003 \end{aligned}$$

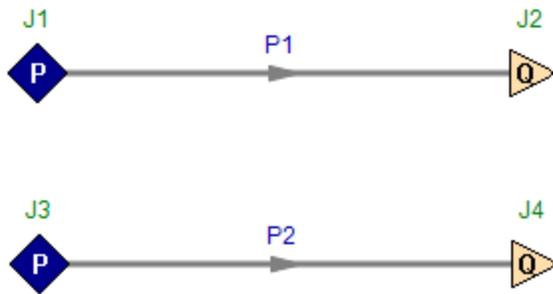
so that, from Figure 6.18,

$$\begin{aligned} f &= 0.027 \\ p_2 - p_1 &= -\frac{1}{2} \rho \bar{V}^2 \frac{fL}{D_h} \\ &= -\frac{1}{2} (997 \text{ kg/m}^3)(1.0)^2 \text{ m}^2/\text{s}^2 \frac{0.027 \times 100 \text{ m}}{0.10 \text{ m}} \\ &= \underline{-13.46 \text{ kPa}} \end{aligned}$$

The discussion of the friction factor has dealt solely with fully developed flow. At a pipe inlet, however, the flow is not fully developed. As can be seen in Figure 6.9, the velocity gradient at the wall and resultant shear stress are actually greater near the inlet. The problem of trying to predict an equivalent friction coefficient for flow near an inlet and its variation with axial distance is extremely complex. Fortunately, it has been shown experimentally that the local friction factor attains its fully developed value roughly 10 diameters from the inlet for turbulent flow; a somewhat greater length is required for laminar flow. It can be seen, then, that any inlet effect can be neglected

View Verification Case 45 Model

[Verification Case 45](#)



Verification Case 46

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify46.fth

REFERENCE: James John, William Haberman, Introduction to Fluid Mechanics, 2nd Ed., 1980, Prentice-Hall, Page 176-178, Example 6.4

FLUID: Water at 20 deg. C

ASSUMPTIONS: Use the K factors provided

RESULTS:

Parameter	John & Haberman	AFT Fathom
Pressure at discharge (kPa gage)	116	115.0

DISCUSSION:

The K factors used by John & Haberman differ from Crane's and those in the AFT Fathom library. The model therefore uses the same K factors as used in the problem solution. In the problem solution, the K factor for contraction area change is applied to the 7.5 cm pipe, while AFT Fathom applies it to the 15 cm pipe. It therefore needed to be multiplied by the area ratio squared, which is 16.

The pipe lengths between fittings were not specified, and do not affect the results. The AFT Fathom assumed pipe lengths, with the total pipe length adding up to 130 m.

Results disagree slightly because John & Haberman read friction factors from a chart, which is less accurate than AFT Fathom's correlation based method. The friction factor for the 7.5 cm pipe was determined to be 0.027 by John & Haberman, and AFT Fathom calculated it as 0.028 based on the Colebrook-White method.

The resulting 115.0 kPa (g) pressure is obtained in AFT Fathom by looking at the exit pressure from the Assigned Flow junction at J8.

[List of All Verification Models](#)

Verification Case 46 Problem Statement

[Verification Case 46](#)

James John, William Haberman, Introduction to Fluid Mechanics, 2nd Ed., 1980, Prentice-Hall, Page 176-178, Example 6.4

[John and Haberman Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

176

Pipe Flow

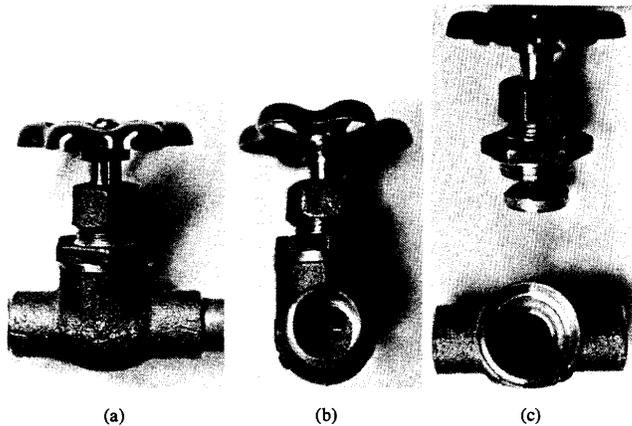


Figure 6.30. Globe valve.

6.5 PIPING SYSTEMS

We are now in a position to work through several examples of complete piping systems. Three basic types of problems are encountered. First, for a given piping system and flow rate, it might be required to find the pressure drop. Second, for a given system and pressure drop, the flow rate might be required. Third, for a given flow and pressure drop, it might be required to design the system, that is, to determine the necessary pipe diameter. In the following material, an example of each type of problem is presented.

EXAMPLE 6.4. For the piping system shown in Figure 6.31 determine the pressure p_2 . There are 100 m of 15-cm-I.D. cast iron pipe and 30 m of 7.5-cm-I.D. cast iron pipe. Water flow rate is 0.01 m³/s, the water temperature is 20°C, and the gage pressure at 1 is 250 kPa.

Solution: First write the modified Bernoulli equation (6.7) between 1 and 2, including minor losses:

$$\frac{p_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \left(\frac{fL}{D} \frac{V^2}{2} \right)_{15\text{-cm pipe}} + \left(\sum K \frac{V^2}{2} \right)_{15\text{-cm pipe}} + \left(\frac{fL}{D} \frac{V^2}{2} \right)_{7.5\text{-cm pipe}} + \left(\sum K \frac{V^2}{2} \right)_{7.5\text{-cm pipe}} + \frac{V_2^2}{2} + gz_2 + \frac{p_2}{\rho}$$

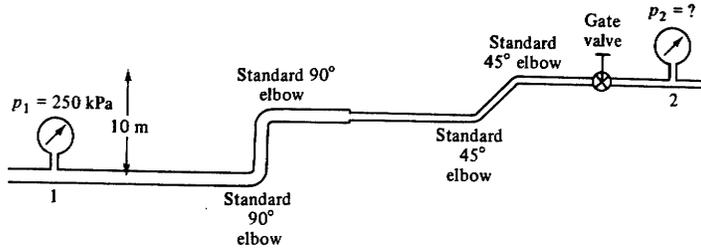


Figure 6.31

The velocity in the 15-cm pipe is

$$V_{15} = \frac{Q}{A} = \frac{0.01 \text{ m}^3/\text{s}}{(\pi/4)(0.15)^2 \text{ m}^2} = 0.5659 \text{ m/s}$$

and in the 7.5-cm pipe it is

$$V_{7.5} = \frac{Q}{A} = \frac{0.01}{(\pi/4)(0.075)^2 \text{ m}^2} = 2.264 \text{ m/s}$$

At 20°C, for water, $\nu = 1.00 \times 10^{-6} \text{ m}^2/\text{s}$, so that

$$Re_{15} = \frac{(0.5659 \text{ m/s})(0.15 \text{ m})}{1.00 \times 10^{-6} \text{ m}^2/\text{s}} = 8.489 \times 10^4$$

$$Re_{7.5} = \frac{(2.264 \text{ m/s})(0.075 \text{ m})}{1.00 \times 10^{-6} \text{ m}^2/\text{s}} = 1.698 \times 10^5$$

Also,

$$\left(\frac{\epsilon}{D}\right)_{15} = \frac{0.26 \times 10^{-3} \text{ m}}{0.15 \text{ m}} = 0.00173$$

$$\left(\frac{\epsilon}{D}\right)_{7.5} = \frac{0.26 \times 10^{-3} \text{ m}}{0.075 \text{ m}} = 0.00346$$

From the Moody diagram, we obtain

$$f_{15} = 0.025$$

$$f_{7.5} = 0.027$$

The minor losses in the 15-cm pipe are given by

$$(\sum K)_{15} = \underbrace{0.75}_{\text{standard elbow}} + \underbrace{0.75}_{\text{standard elbow}} = 1.50$$

For the 7.5-cm pipe,

$$(\sum K)_{7.5} = \underbrace{0.33}_{\text{contraction}} + \underbrace{0.35}_{\text{elbow}} + \underbrace{0.35}_{\text{elbow}} + \underbrace{0.20}_{\text{gate valve}} = 1.23$$

Substituting into the Bernoulli equation, we have

$$\begin{aligned} \frac{p_2 - p_1}{\rho} &= g(z_1 - z_2) + \frac{V_1^2}{2} \left(1 - \frac{fL}{D} - \Sigma K \right)_{15} - \frac{V_2^2}{2} \left(1 + \frac{fL}{D} + \Sigma K \right)_{7.5} \\ &= 9.81 \text{ m/s}^2 (-10 \text{ m}) + \left(\frac{0.5659^2}{2} \text{ m}^2/\text{s}^2 \right) \left(1 - \frac{0.025 \times 100}{0.15} - 1.50 \right) \\ &\quad - \left(\frac{2.264^2}{2} \text{ m}^2/\text{s}^2 \right) \left(1 + \frac{0.027 \times 30}{0.075} + 1.23 \right) \\ &= -98.1 \text{ m}^2/\text{s}^2 + (0.1601 \text{ m}^2/\text{s}^2)(-17.17) - (2.563 \text{ m}^2/\text{s}^2)(13.03) \\ &= -98.1 \text{ m}^2/\text{s}^2 - 2.749 \text{ m}^2/\text{s}^2 - 33.40 \text{ m}^2/\text{s}^2 \\ &= -134.2 \text{ m}^2/\text{s}^2 \end{aligned}$$

or

$$\begin{aligned} p_2 - p_1 &= -(998.3 \text{ kg/m}^3)(134.2 \text{ m}^2/\text{s}^2) \\ &= -134 \text{ kPa} \end{aligned}$$

or

$$\begin{aligned} p_2 &= 250 - 134 \\ &= \underline{116 \text{ kPa}} \text{ (gage)} \end{aligned}$$

EXAMPLE 6.5. For the piping system shown in Figure 6.32, determine the water flow rate. The water temperature is 25°C, with all pipe asphalted cast iron. There are 250 m of 10-cm-I.D. pipe.

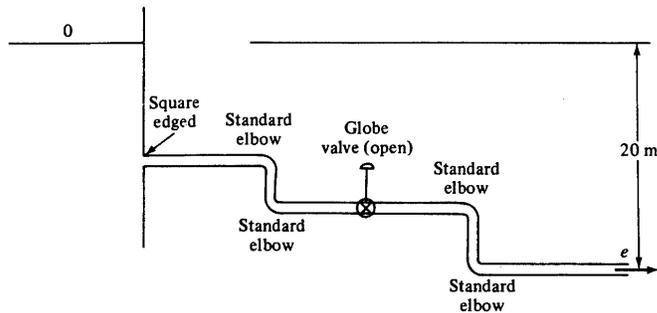


Figure 6.32

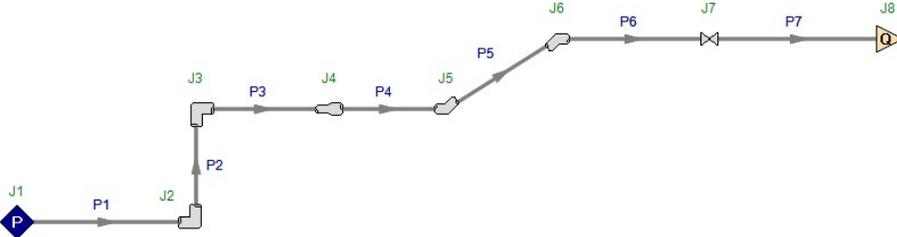
Solution: Write the modified Bernoulli equation between the reservoir surface and pipe outlet:

$$\frac{p_0}{\rho} + gz_0 + \frac{V_0^2}{2} = \frac{fL}{D} \frac{V^2}{2} + \Sigma K \frac{V^2}{2} + \frac{p_e}{\rho} + gz_e + \frac{V_e^2}{2}$$

where $p_0 = p_e =$ atmospheric pressure; $z_0 - z_e = 20 \text{ m}$; and $V_0 = 0$. For this case, with flow velocity the unknown, Reynolds numbers cannot be directly

View Verification Case 46 Model

[Verification Case 46](#)



Verification Case 47

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify47.fth

REFERENCE: James John, William Haberman, Introduction to Fluid Mechanics, 2nd Ed., 1980, Prentice-Hall, Page 178-179, Example 6.5

FLUID: Water at 20 deg. C (incorrectly stated as 25 deg. C)

ASSUMPTIONS: Use the K factors provided

RESULTS:

Parameter	John & Haberman	AFT Fathom
Flow rate through pipe line (liter/sec)	13.55*	19.33
Flow rate through pipe line (liter/sec)	19.17**	19.33

(*) Calculation error in text explained below

(**) Corrected flow rate calculation as explained below

DISCUSSION:

The K factors used by John & Haberman differ from Crane's and those in the AFT Fathom library. The model therefore uses the same K factors as used in the problem solution. The pipe lengths between fittings were not specified, and do not affect the results. The AFT Fathom assumed pipe lengths, with the total pipe length adding up to 250 m.

The problem statement is for water at 25 deg. C, but when the Reynolds number is calculated on page 179 the kinematic viscosity is that at 20 deg. C. Therefore the AFT Fathom model uses 20 deg. C water.

The results disagree significantly. A review of John & Haberman page 179 shows a calculation error. The text says the following:

$$\frac{V_e^2}{2} = \frac{9.81(20)}{55 + 9.9 + 1}$$

Or

$$V_e = 1.725 \text{ m/s}$$

which is incorrect. Solving the above equation by a hand calculator shows that

$$V_e = 2.44 \text{ m/s}$$

If the incorrect velocity is used the final flow rate in John & Haberman is significantly different that Fathom.

$$\begin{aligned}Q &= AV \\&= \left[\frac{\pi}{4} (0.10)^2 m^2 \right] 1.725 m/s \\&= 0.01355 m^3/s \\&= 13.55 l/s\end{aligned}$$

When using the corrected velocity the final flow rate in John & Haberman is much closer to the Fathom result.

$$\begin{aligned}Q &= AV \\&= \left[\frac{\pi}{4} (0.10)^2 m^2 \right] 2.44 m/s \\&= 0.01917 m^3/s \\&= 19.17 l/s\end{aligned}$$

Using the correct velocity results in the correct flow rate, which agrees closely with AFT Fathom.

[List of All Verification Models](#)

Verification Case 47 Problem Statement

Verification Case 47

James John, William Haberman, Introduction to Fluid Mechanics, 2nd Ed., 1980, Prentice-Hall, Page 178-179, Example 6.5

John and Haberman Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

178

Pipe Flow

Substituting into the Bernoulli equation, we have

$$\begin{aligned} \frac{p_2 - p_1}{\rho} &= g(z_1 - z_2) + \frac{V_1^2}{2} \left(1 - \frac{fL}{D} - \Sigma K \right)_{15} - \frac{V_2^2}{2} \left(1 + \frac{fL}{D} + \Sigma K \right)_{7.5} \\ &= 9.81 \text{ m/s}^2 (-10 \text{ m}) + \left(\frac{0.5659^2}{2} \text{ m}^2/\text{s}^2 \right) \left(1 - \frac{0.025 \times 100}{0.15} - 1.50 \right) \\ &\quad - \left(\frac{2.264^2}{2} \text{ m}^2/\text{s}^2 \right) \left(1 + \frac{0.027 \times 30}{0.075} + 1.23 \right) \\ &= -98.1 \text{ m}^2/\text{s}^2 + (0.1601 \text{ m}^2/\text{s}^2)(-17.17) - (2.563 \text{ m}^2/\text{s}^2)(13.03) \\ &= -98.1 \text{ m}^2/\text{s}^2 - 2.749 \text{ m}^2/\text{s}^2 - 33.40 \text{ m}^2/\text{s}^2 \\ &= -134.2 \text{ m}^2/\text{s}^2 \end{aligned}$$

or

$$\begin{aligned} p_2 - p_1 &= -(998.3 \text{ kg/m}^3)(134.2 \text{ m}^2/\text{s}^2) \\ &= -134 \text{ kPa} \end{aligned}$$

or

$$\begin{aligned} p_2 &= 250 - 134 \\ &= \underline{116 \text{ kPa (gage)}} \end{aligned}$$

EXAMPLE 6.5. For the piping system shown in Figure 6.32, determine the water flow rate. The water temperature is 25°C, with all pipe asphalted cast iron. There are 250 m of 10-cm-I.D. pipe.

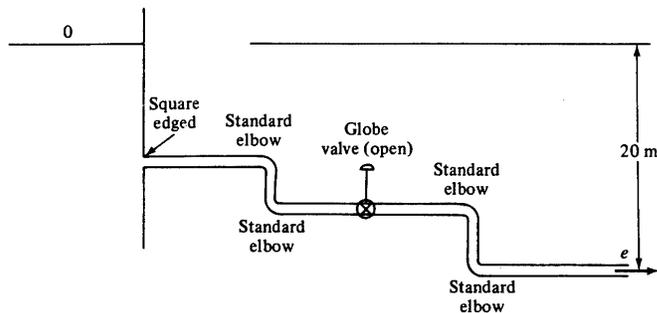


Figure 6.32

Solution: Write the modified Bernoulli equation between the reservoir surface and pipe outlet:

$$\frac{p_0}{\rho} + gz_0 + \frac{V_0^2}{2} = \frac{fL}{D} \frac{V^2}{2} + \Sigma K \frac{V^2}{2} + \frac{p_e}{\rho} + gz_e + \frac{V_e^2}{2}$$

where $p_0 = p_e =$ atmospheric pressure; $z_0 - z_e = 20 \text{ m}$; and $V_0 = 0$. For this case, with flow velocity the unknown, Reynolds numbers cannot be directly

determined; a trial-and-error solution is called for. As a first trial, assume $f = 0.02$.

$$(9.81 \text{ m/s}^2)(20 \text{ m}) = \left[\frac{0.02 \times 250}{0.10} + \left(\underset{\substack{\text{square-} \\ \text{edged} \\ \text{inlet}}}{0.5} + \underset{\substack{4 \text{ elbows}}}{3.0} + \underset{\substack{\text{globe} \\ \text{valve}}}{6.4} \right) + 1 \right] \frac{V_e^2}{2}$$

$$\frac{V_e^2}{2} = \frac{9.81(20)}{50 + 9.9 + 1} \text{ m}^2/\text{s}^2$$

or

$$V_e = 1.795 \text{ m/s}$$

For this velocity,

$$\text{Re} = \frac{VD}{\nu} = \frac{(1.795 \text{ m/s})(0.10 \text{ m})}{1.00 \times 10^{-6} \text{ m}^2/\text{s}} = 1.795 \times 10^5$$

$$\frac{\epsilon}{D} = \frac{0.12 \times 10^{-3} \text{ m}}{0.10 \text{ m}} = 1.2 \times 10^{-3}$$

From the Moody diagram, we obtain $f = 0.022$. Since this value does not agree with our initial trial, we shall now make a second trial, starting with $f = 0.022$.

$$\frac{fL}{D} = \frac{0.022 \times 250}{0.10} = 55$$

$$\frac{V_e^2}{2} = \frac{9.81(20)}{55 + 9.9 + 1}$$

or

$$V_e = 1.725 \text{ m/s}$$

For this velocity,

$$\text{Re} = \frac{VD}{\nu} = \frac{1.725 \times 0.10}{10^{-6}} = 1.725 \times 10^5$$

From the Moody diagram, we find $f = 0.022$. It can be seen that generally we are able to converge on the correct answer quite rapidly; no more than two trials are required, as a rule. The water flow rate is

$$Q = AV$$

$$= \left[\frac{\pi}{4} (0.10)^2 \text{ m}^2 \right] 1.725 \text{ m/s}$$

$$= 0.01355 \text{ m}^3/\text{s}$$

$$= \underline{13.55 \text{ l/s}}$$

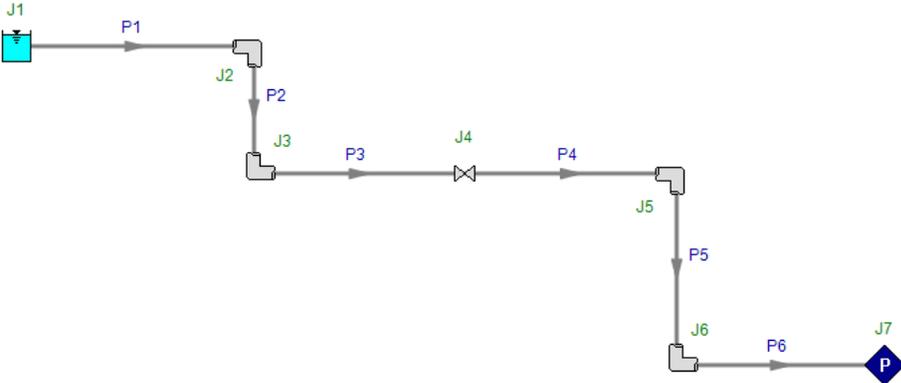
EXAMPLE 6.6. A pump is to be used to supply 5 liters per second of water from a reservoir to a point 400 m from the reservoir at the same level as the reservoir surface. Determine the minimum-size cast iron pipe required. The water temperature is 15°C; assume that minor losses can be neglected. (See Figure 6.33.) The pump supplies 50 kW of power to the water flow.

Solution: From (6.8) we can express the pump work in terms of reservoir surface and outlet conditions:

$$-\frac{1}{\dot{m}} \frac{dW_p}{dt} = \frac{p_e - p_0}{\rho} + g(z_e - z_0) + \frac{fL}{D} \frac{V^2}{2} + \frac{V_e^2}{2}$$

View Verification Case 47 Model

[Verification Case 47](#)



Verification Case 48

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify48.fth

REFERENCE: James John, William Haberman, Introduction to Fluid Mechanics, 2nd Ed., 1980, Prentice-Hall, Page 184-186, Example 6.8

FLUID: Water at 20 deg. C

ASSUMPTIONS: All pipes at same elevation. Common exit pressure is 1 atm.

RESULTS:

Parameter	John & Haberman	AFT Fathom
Flow rate through pipe 1 (m3/sec)	0.095	0.0943
Flow rate through pipe 2 (m3/sec)	0.105	0.1057

DISCUSSION:

The problem statement does not have any pressure information, because it is not required if the only purpose is to calculate flow rates. However, AFT Fathom requires at least one pressure boundary because AFT Fathom offers pressure information in the output for all systems. Therefore, a pressure of 1 atm was assumed for the common junction at the exit of the system.

[List of All Verification Models](#)

Verification Case 48 Problem Statement

Verification Case 48

James John, William Haberman, Introduction to Fluid Mechanics, 2nd Ed., 1980, Prentice-Hall, Page 184-186, Example 6.8

John and Haberman Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

184

Pipe Flow

or, with $f = 0.02$,

$$\frac{\bar{V}^2}{D_h} = \frac{5.23(2)}{1.06(0.02)} = 493.7 \text{ m/s}^2$$

Now $Q = \bar{V}A = \bar{V}w(0.08)$.

$$\bar{V} = \frac{5}{60(0.08)w} = \frac{1.042}{w}$$

and

$$D_h = \frac{2dw}{(d+w)} = 2 \frac{0.08w}{0.08+w}$$

Hence

$$\frac{\bar{V}^2}{D_h} = \frac{(1.042)^2(w+0.08)}{w^3 \cdot 0.16} = 6.782 \frac{w+0.08}{w^3}$$

Thus

$$493.7 = 6.782 \frac{w+0.08}{w^3} \quad \text{or} \quad 72.75w^3 = w + 0.08$$

Using a trial-and-error solution for w ,

w	$72.75w^3$	$w + 0.08$
0.1	0.073	0.18
0.15	0.246	0.23
0.14	0.200	0.22
0.145	0.222	0.225
0.146	0.226	0.226

we obtain $w = 146 \text{ mm}$.

EXAMPLE 6.8. Twelve cubic meters per minute of water flow into a horizontal pipe network consisting of two cast iron pipe branches (Figure 6.37). The first branch is 100 m long with a 200-mm pipe diameter, and the second branch is 200 m long with a 250-mm pipe diameter and a half-open gate valve. Determine the flow through each pipe and the pressure drop across the branches. Water temperature is 20°C.

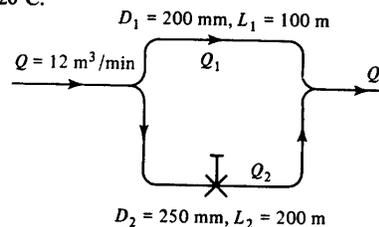


Figure 6.37

Solution: From (6.7) and Section 6.5, the pressure drop in each branch is given by:

$$\Delta p = f \frac{L}{D} \frac{V^2}{2} + \sum K \frac{V^2}{2}$$

The velocity in each branch can be expressed as $V = Q/(\pi/4)D^2$, so that, for each branch:

$$\Delta p = \frac{16}{\pi^2} \left(\frac{fL}{D} + \sum K \right) \frac{Q^2}{D^5}$$

But $\Delta p_1 = \Delta p_2$ and $\sum K_1 = 0$; therefore, the ratio of volume flow rates is

$$\frac{Q_1}{Q_2} = \left(\frac{D_1}{D_2} \right)^2 \sqrt{\frac{f_2 L_2 / D_2 + \sum K_2}{f_1 L_1 / D_1}}$$

As a first trial, assume $f_1 = f_2 = 0.020$. From Table 6.3, $K_2 = 4.5$ for a half-open gate valve; hence

$$\frac{f_2 L_2}{D_2} + \sum K_2 = 0.020 \frac{200}{0.250} + 4.5 = 20.5$$

and

$$\frac{f_1 L_1}{D_1} = 0.020 \frac{100}{0.2} = 10$$

Therefore,

$$\frac{Q_1}{Q_2} = \left(\frac{0.200}{0.250} \right)^2 \sqrt{\frac{20.5}{10}} = 0.916$$

$$\text{Total flow } Q = Q_1 + Q_2, \quad \text{or} \quad \frac{Q}{Q_2} = \frac{Q_1}{Q_2} + 1 = 1.916$$

and

$$Q_2 = \frac{1.2}{1.916} = 0.1044 \text{ m}^3/\text{s}$$

with $Q_1 = 0.2 - 0.1044 = 0.0956 \text{ m}^3/\text{s}$

Then

$$V_2 = \frac{0.1044}{(\pi/4)(0.250)^2} = 2.13 \text{ m/s}$$

From Appendix A, $\nu = 1.00 \times 10^{-6} \text{ m}^2/\text{s}$ at 20°C for water, so

$$\text{Re}_2 = \frac{V_2 D_2}{\nu} = \frac{(2.13 \text{ m/s})(0.250 \text{ m})}{1.00 \times 10^{-6} \text{ m}^2/\text{s}} = 5.32 \times 10^5$$

From Table 6.1, $\epsilon = 0.26 \text{ mm}$ for cast iron pipe; hence,

$$\frac{\epsilon}{D_2} = \frac{0.00026}{0.250} = 0.001$$

Hence, from Figure 6.18, we find $f_2 = 0.02$, which was our initial assumption.

Now,

$$\begin{aligned} \Delta p_2 &= \frac{\rho V_2^2}{2} \left(\frac{f_2 L_2}{D} + K_2 \right) = \frac{(998.3 \text{ kg/m}^3)(2.13^2 \text{ m}^2/\text{s}^2)}{2} 20.5 \\ &= 46.5 \text{ kPa} \end{aligned}$$

where $\rho = 998.3 \text{ kg/m}^3$ at 20°C . Further,

$$V_1 = \frac{0.0956 \text{ m}^3/\text{s}}{(\pi/4)(0.200)^2 \text{ m}^2} = 3.04 \text{ m/s}$$

and

$$\text{Re}_1 = \frac{(3.04 \text{ m/s})(0.2 \text{ m})}{1 \times 10^{-6} \text{ m}^2/\text{s}} = 6.09 \times 10^5$$

$$\frac{\epsilon}{D_1} = \frac{0.00026 \text{ m}}{0.200 \text{ m}} = 0.0013$$

From Figure 6.17, $f_1 = 0.0208$, and hence

$$\begin{aligned} \Delta p_1 &= \frac{\rho V_1^2 f L_1}{2 D_1} = \frac{(998.3 \text{ kg/m}^3)(3.04^2 \text{ m}^2/\text{s}^2) 0.0208 (100 \text{ m})}{2 \times 0.2 \text{ m}} \\ &= 48.0 \text{ kPa} \end{aligned}$$

as compared to $\Delta p_2 = 46.5 \text{ kPa}$.

For our second trial, we increase Q_2 and decrease Q_1 to balance the pressure drops. Let $Q_2 = 0.105 \text{ m}^3/\text{s}$ and $Q_1 = 0.095 \text{ m}^3/\text{s}$. Then

$$V_2 = \frac{0.105 \text{ m}^3/\text{s}}{(\pi/4)(0.25)^2} = 2.14 \text{ m/s}$$

$$\text{Re}_2 = \frac{(2.14 \text{ m/s})(0.25 \text{ m})}{1 \times 10^{-6} \text{ m}^2/\text{s}} = 5.34 \times 10^5$$

Therefore,

$$f_2 = 0.02 \quad \text{and} \quad \Delta p_2 = 47.0 \text{ kPa}$$

$$V_1 = \frac{0.095 \text{ m}^3/\text{s}}{(\pi/4)(0.200)^2 \text{ m}^2} = 3.02 \text{ m/s}$$

$$\text{Re}_1 = 6.05 \times 10^5$$

$$f_1 = 0.0208, \quad \Delta p_1 = 47.3 \text{ kPa}$$

Thus the solution is

$$Q_1 = 0.095 \text{ m}^3/\text{s}, \quad Q_2 = 0.105 \text{ m}^3/\text{s}$$

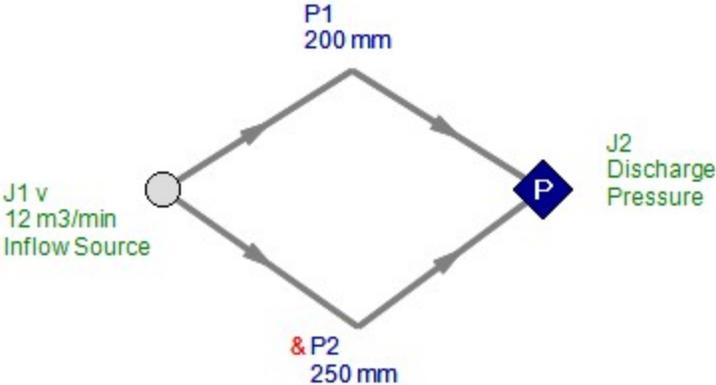
with a pressure drop Δp of 47 kPa. Note that, although branch 2 has a larger diameter than branch 1, it carries less flow due to the presence of a valve and due to its greater length.

PROBLEMS

- 6.1. Water at 10°C flows steadily through a 5-cm-diameter pipe. The pressure drop in a 3-m length of pipe is found to be 750 Pa. Compute the wall shear stress.
- 6.2. Air at 20°C flows steadily through a rectangular duct, 5 cm by 10 cm with a velocity of 2 m/s. Is the flow laminar or turbulent?
- 6.3. For Problem 6.2, compute the pressure drop per meter of duct, with $f = 0.02$, $\epsilon = 0.10 \text{ mm}$.

View Verification Case 48 Model

[Verification Case 48](#)



Verification Case 49

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify49.fth

REFERENCE: William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Page 153-154, Example 5.2

FLUID: Water

ASSUMPTIONS: Water assumed to be 20 deg. C

RESULTS:

Parameter	Janna	AFT Fathom
Pressure drop in pipe (kPa)	347.1	347.1

DISCUSSION:

The friction factor for this problem was defined explicitly based on the problem statement rather than using the iterative Colebrook-White equation.

[List of All Verification Models](#)

Verification Case 49 Problem Statement

Verification Case 49

William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Page 153-154, Example 5.2

Janna Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

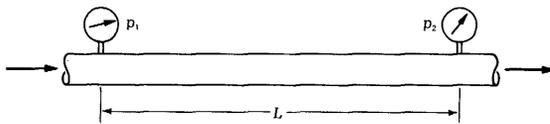


Figure 5.10. Flow in a constant-diameter pipe.

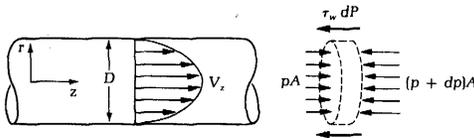


Figure 5.11. Control volume of a system: flow in a duct.

Integrating this expression from points 1 to 2 a distance L apart in the conduit yields

$$p_2 - p_1 = -\frac{\rho V^2 f L}{2g_c D} \quad (5.7)$$

Equation 5.7 gives the pressure drop in the duct due to friction. With this equation, the Bernoulli equation can now be rewritten as

$$\frac{p_1 g_c}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{p_2 g_c}{\rho g} + \frac{V_2^2}{2g} + z_2 + \sum \frac{fL}{D} \frac{V^2}{2g} \quad (5.8)$$

which now takes wall friction into account. The summation indicates that if several pipes of different diameter are connected in series, the total friction drop is due to the combined effect of them all. The frictional effect is manifested in either a heat loss from the fluid or a gain in internal energy of the fluid.

EXAMPLE 5.2

A 2-nominal pipe is inclined at an angle of 30° with the horizontal and conveys $0.001 \text{ m}^3/\text{s}$ of water uphill. Determine the pressure drop in the pipe if it is 70 m long (see Figure 5.12). Take the friction factor f to be 0.03.

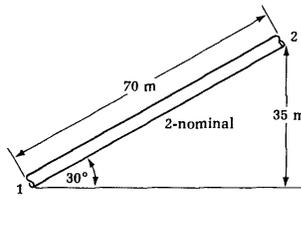


Figure 5.12. Sketch for Example 5.2.

SOLUTION

The Bernoulli equation with the friction term applies:

$$\frac{p_1 g_c}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{p_2 g_c}{\rho g} + \frac{V_2^2}{2g} + z_2 + \sum \frac{fL}{D} \frac{V^2}{2g}$$

or

$$p_1 - p_2 = \frac{\rho}{2g_c} (V_2^2 - V_1^2) + \frac{\rho g}{g_c} (z_2 - z_1) + \frac{\rho V^2 fL}{2g_c D}$$

From continuity, $A_1 V_1 = A_2 V_2$ or $V_1 = V_2$. If z_1 is our reference point, then $z_1 = 0$ and $z_2 = 35$ m. From Appendix Table C-1, the inside diameter of 2-nominal pipe is 5.252 cm for schedule 40. (Because a schedule was not specified in the problem statement, assume the standard.) The average velocity in the pipe is

$$V = \frac{Q}{A} = \frac{0.001}{0.0022} = 0.455 \text{ m/s}$$

where the area A for schedule 40, 2-nominal pipe is obtained also from Appendix Table C-1. By substitution into the Bernoulli equation, we obtain

$$\begin{aligned} p_1 - p_2 &= 0 + 1000(9.8)(35 - 0) + \frac{1000(0.455)^2}{2} \frac{(0.03)(70)}{0.05252} \\ &= 343\,000 + 4\,139 = 347\,139 \text{ Pa} \end{aligned}$$

$$\underline{p_1 - p_2 = 347.1 \text{ kPa}}$$

5.5 / The Friction Factor and Pipe Roughness

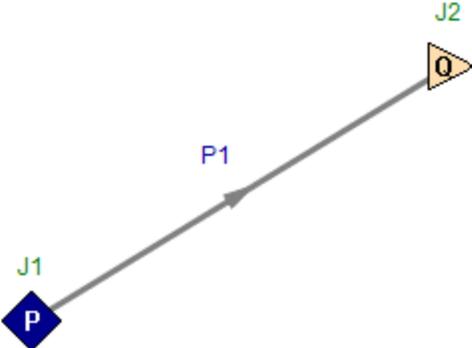
The preceding section introduced the concept of friction and a friction factor. In this section we will discuss this concept further for two flow regimes—laminar and turbulent flow. Because the friction factor is influenced by the fluid velocity, we begin with a velocity distribution for circular pipe flow and later extend the results to other cross sections.

Consider laminar flow in a circular pipe as shown in Figure 5.13. Cylindrical coordinates are chosen with z as the axial direction. A control volume is selected as shown, and in applying the momentum equation we obtain

$$\Sigma F_z = \iint_{CS} V_z \frac{\rho}{g_c} V_n dA$$

View Verification Case 49 Model

[Verification Case 49](#)



Verification Case 50

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify50.fth

REFERENCE: William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Page 166-167, Example 5.3

FLUID: Water at 22 deg. C

ASSUMPTIONS: N/A

RESULTS:

Parameter	John & Haberman	AFT Fathom
Head loss in pipe (meters)	0.744	0.759
Pressure drop in pipe (kPa)	7.279	7.434

DISCUSSION:

Results disagree slightly because Janna reads friction factors from a chart, which is less accurate than AFT Fathom's correlation based method. The friction factor was determined to be 0.022 by Janna, and AFT Fathom calculated it as 0.02244 based on the Colebrook-White method, which is 2% higher.

[List of All Verification Models](#)

Verification Case 50 Problem Statement

[Verification Case 50](#)

William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Page 166-167, Example 5.3

[Janna Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

166 Chapter 5 / Flow in Closed Conduits

In simple piping systems there are six variables that enter the problem:

L = pipe length

D = pipe diameter or hydraulic diameter of conduit

$\nu = \frac{\mu g_c}{\rho}$ = kinematic viscosity of fluid

ϵ = wall roughness

Q = volume flow rate

$h_f = \frac{V^2 f L}{2g D}$ = head loss due to friction

Of these variables, L , ν , and ϵ are generally known. Thus three types of problems are commonly encountered:

1. Given L , ν , ϵ , Q , and D , find h_f .
2. Given L , ν , ϵ , D , and h_f , find Q .
3. Given L , ν , ϵ , h_f , and Q , find D .

The solution technique for each of these problems is illustrated by example.

EXAMPLE 5.3

Water at $0.02 \text{ m}^3/\text{s}$ flows through 350 m of 8-nominal cast iron pipe. Determine the head loss if the water temperature is 22°C .

SOLUTION

We know that $Q = 0.02 \text{ m}^3/\text{s}$ and $L = 350 \text{ m}$. From Appendix Table C-1, $D = 20.27 \text{ cm}$ for schedule 40 pipe and $A = 322.70 \text{ cm}^2$. From Appendix Table A-4, $\nu = 0.9548 \times 10^{-3}/997.8 = 9.569 \times 10^{-7} \text{ m}^2/\text{s}$. By definition,

$$V = \frac{Q}{A} = \frac{0.02}{0.032270} = 0.62 \text{ m/s}$$

so

$$\text{Re} = \frac{VD}{\nu} = \frac{0.62(0.2027)}{9.569 \times 10^{-7}} = 1.312 \times 10^5$$

Also, from Table 5.2,

$$\epsilon = 0.025 \text{ cm}$$

Thus

$$\frac{\epsilon}{D} = \frac{0.025}{20.27} = 0.0012$$

On the $\epsilon/D = 0.001$ line of Figure 5.15a, follow to the left from the turbulence zone until the $Re = 1.312 \times 10^5$ point is reached. At this intersection, read

$$f = 0.022$$

By substitution into the equation for head loss, we obtain

$$h_f = \frac{fL}{D} \frac{V^2}{2g} = \frac{0.022(350)}{0.2027} \frac{(0.62)^2}{2(9.81)}$$

$$h_f = \underline{0.744 \text{ m of water}}$$

Further,

$$\Delta p = \frac{\rho g}{g_c} h_f = 997(9.81)(0.744) = 7\,279 \text{ N/m}^2$$

or

$$\Delta p = \underline{7.279 \text{ kPa}}$$

The method and calculations in the preceding example are quite straightforward. In the following example, Q is the unknown and therefore V and f are unknowns. A trial and error solution is required, but the technique is simple.

EXAMPLE 5.4

Benzene flows through a schedule 80 12-nominal wrought iron pipe. The pressure drop measured at points 1200 ft apart is 15 psi. Determine the volume flow rate through the pipe.

SOLUTION

We know that $L = 1200$ ft and $\Delta p = 15$ psi. From Appendix Table A-5, $\rho = 0.876(62.4) = 54.7$ lbm/ft³ and $\mu_{g_c} = 4.04 \times 10^{-4}$ lbm/ft-s for benzene. From Appendix Table C-1, $D = 0.9478$ ft for schedule 80 12-nominal pipe. From Table 5.2, $\epsilon = 0.00015$ ft. The head loss is

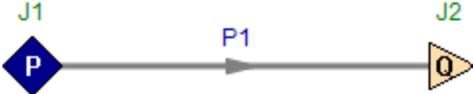
$$h_f = \frac{\Delta p}{\rho} \frac{g_c}{g} = \frac{15(144)}{54.7} \frac{32.2}{32.2} = 39.49 \text{ ft of benzene}$$

The relative roughness is

$$\frac{\epsilon}{D} = \frac{0.00015}{0.9478} = 0.00016$$

View Verification Case 50 Model

[Verification Case 50](#)



Verification Case 51

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify51.fth

REFERENCE: William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Page 167-168, Example 5.4

FLUID: Benzene at 77 deg. F

ASSUMPTIONS: N/A

RESULTS:

Parameter	Janna	AFT Fathom
Flow rate (ft ³ /sec)	8.51	8.49

DISCUSSION:

Although Janna only performed one iteration for friction factor while AFT Fathom performed further iterations, the results were still very similar due to the small difference in calculated friction factors.

[List of All Verification Models](#)

Verification Case 51 Problem Statement

[Verification Case 51](#)

William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Page 167-168, Example 5.4

[Janna Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

On the $\epsilon/D = 0.001$ line of Figure 5.15a, follow to the left from the turbulence zone until the $Re = 1.312 \times 10^5$ point is reached. At this intersection, read

$$f = 0.022$$

By substitution into the equation for head loss, we obtain

$$h_f = \frac{fL}{D} \frac{V^2}{2g} = \frac{0.022(350)}{0.2027} \frac{(0.62)^2}{2(9.81)}$$

$$\underline{h_f = 0.744 \text{ m of water}}$$

Further,

$$\Delta p = \frac{\rho g}{g_c} h_f = 997(9.81)(0.744) = 7279 \text{ N/m}^2$$

or

$$\underline{\Delta p = 7.279 \text{ kPa}}$$

The method and calculations in the preceding example are quite straightforward. In the following example, Q is the unknown and therefore V and f are unknowns. A trial and error solution is required, but the technique is simple.

EXAMPLE 5.4

Benzene flows through a schedule 80 12-nominal wrought iron pipe. The pressure drop measured at points 1200 ft apart is 15 psi. Determine the volume flow rate through the pipe.

SOLUTION

We know that $L = 1200$ ft and $\Delta p = 15$ psi. From Appendix Table A-5, $\rho = 0.876(62.4) = 54.7$ lbm/ft³ and $\mu_{g_c} = 4.04 \times 10^{-4}$ lbm/ft-s for benzene. From Appendix Table C-1, $D = 0.9478$ ft for schedule 80 12-nominal pipe. From Table 5.2, $\epsilon = 0.00015$ ft. The head loss is

$$h_f = \frac{\Delta p}{\rho} \frac{g_c}{g} = \frac{15(144)}{54.7} \frac{32.2}{32.2} = 39.49 \text{ ft of benzene}$$

The relative roughness is

$$\frac{\epsilon}{D} = \frac{0.00015}{0.9478} = 0.00016$$

Using Figure 5.15a (or b), as a first trial estimate the friction factor for this flow to be that for completely turbulent flow; thus $f = 0.0138$. By definition,

$$h_f = \frac{fL}{D} \frac{V^2}{2g}$$

or

$$V = \sqrt{\frac{2gh_f D}{fL}} = \sqrt{\frac{2(32.2)(39.49)(0.9478)}{0.0138(1200)}} = 12.06 \text{ ft/s}$$

Also

$$\text{Re} = \frac{\rho V D}{\mu g_c} = \frac{54.7(12.06)(0.9478)}{4.04 \times 10^{-4}} = 1.55 \times 10^6$$

At $\text{Re} = 1.55 \times 10^6$ and $\epsilon/D = 0.00016$, we find (from Figure 5.15a),

$$f = 0.0139$$

which is close enough to our assumed value. So

$$V = 12.06 \text{ ft/s}$$

and

$$Q = AV = \frac{\pi}{4}(0.9478)^2(12.06)$$

$$Q = 8.51 \text{ ft}^3/\text{s}$$

If the value of f determined did not check with the assumed value, then for our second trial we would begin with $f = 0.0139$. The method converges very rapidly. Seldom are more than two trials necessary.

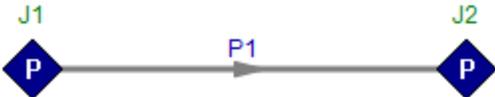
In the next example diameter is unknown. Therefore f and V are also unknown and again a trial and error solution is required. It is convenient to develop equations to streamline the calculations before proceeding, however. By definition,

$$h_f = \frac{fL}{D} \frac{V^2}{2g}$$

Combining with

View Verification Case 51 Model

[Verification Case 51](#)



Verification Case 52

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify52.fth

REFERENCE: William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Page 176-177, Example 5.6

FLUID: Turpentine

ASSUMPTIONS: Use Janna K factors

RESULTS:

Parameter	Janna	AFT Fathom
Static Pressure difference (kPa)	3.012	2.938

DISCUSSION:

The various pipe lengths and intermediate junction elevations are not specified by Janna, so values were assigned to the AFT Fathom model such that the total lengths of 60 m and 22 m resulted, and the end-point elevations were correct.

Results disagree slightly because Janna reads friction factors from a chart, which is less accurate than AFT Fathom's correlation based method.

The pressure drop in AFT Fathom can be obtained by subtracting the exit pressure from the inlet in the Output window junction results table, or in the Junction Deltas table at the top of the Output window. The junction delta was created in the Output Control window.

[List of All Verification Models](#)

Verification Case 52 Problem Statement

[Verification Case 52](#)

William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Page 176-177, Example 5.6

[Janna Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

176 Chapter 5 / Flow in Closed Conduits

EXAMPLE 5.6

In a processing plant, turpentine is piped from tanks to cans that are to be sealed and sold to retail outlets. A portion of the pipeline is sketched in Figure 5.22. There are 60 m of 12-nominal pipe and 22 m of 8-nominal pipe. All elbows are standard and flanged, and the line is made of schedule 80 wrought iron pipe. Determine the pressure drop $p_1 - p_2$ if the volume rate of flow is $0.05 \text{ m}^3/\text{s}$.

SOLUTION

First we write the modified Bernoulli equation from section 1 to 2:

$$\frac{p_1 g_c}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{p_2 g_c}{\rho g} + \frac{V_2^2}{2g} + z_2 + \left(\frac{fL}{D} + \Sigma K \right) \frac{V^2}{2g} \Big|_{12\text{-pipe}} + \left(\frac{fL}{D} + \Sigma K \right) \frac{V^2}{2g} \Big|_{8\text{-pipe}}$$

From Appendix Table C-1 (for 12-nominal and 8-nominal pipe):

$$\begin{aligned} D_{12} &= 28.89 \text{ cm} = 0.289 \text{ m} & A_{12} &= 0.065550 \text{ m}^2 \\ D_8 &= 0.194 \text{ m} & A_8 &= 0.029470 \text{ m}^2 \end{aligned}$$

Therefore

$$V_{12} = \frac{Q}{A_{12}} = \frac{0.05}{0.065550} = 0.762 \text{ m/s}$$

$$V_8 = \frac{Q}{A_8} = \frac{0.05}{0.029470} = 1.697 \text{ m/s}$$

From Appendix Table A-5 for turpentine, $\rho = 870 \text{ kg/m}^3$ and $\mu = 1.375 \times 10^{-3} \text{ N}\cdot\text{s/m}^2$. Thus

$$\text{Re}_{12} = \frac{\rho V D}{\mu g_c} = \frac{870(0.762)(0.289)}{1.375 \times 10^{-3}} = 1.39 \times 10^5$$

$$\text{Re}_8 = \frac{870(1.697)(0.194)}{1.375 \times 10^{-3}} = 2.08 \times 10^5$$

From Table 5.2 for wrought iron, $\epsilon = 0.0046 \text{ cm}$; we find

$$\frac{\epsilon}{D_{12}} = \frac{0.0046}{28.89} = 0.00016$$

$$\frac{\epsilon}{D_8} = \frac{0.0046}{19.37} = 0.00024$$

From the Moody diagram,

$$f_{12} = 0.018$$

$$f_8 = 0.0175$$

The minor losses in the 12-pipe are from one gate valve and four standard elbows:

$$\Sigma K|_{12} = 0.15 + 4(0.31) = 1.39$$

The minor losses in the 8-pipe area are from the contraction ($D_1^2/D_2^2 = 0.45$) and two 45° elbows:

$$\Sigma K|_8 = 0.29 + 2(0.17) = 0.63$$

After rearranging the Bernoulli equation and substituting, we obtain

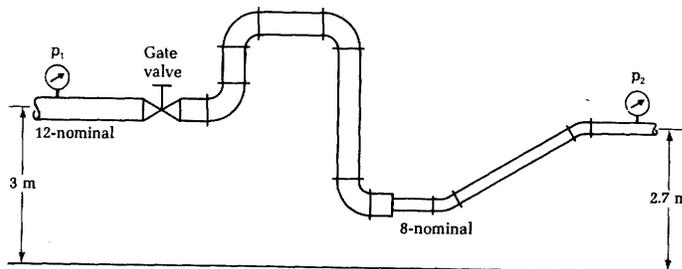
$$\begin{aligned} \frac{p_1 - p_2}{\rho} \frac{\rho}{g} &= \frac{(1.697)^2}{2(9.81)} - \frac{(0.762)^2}{2(9.81)} + (2.7 - 3) \\ &+ \left[\frac{0.018(60)}{0.289} + 1.39 \right] \frac{(0.762)^2}{2(9.81)} \\ &+ \left[\frac{0.0175(22)}{0.194} + 0.63 \right] \frac{(1.697)^2}{2(9.81)} \\ &= 0.1468 - 0.0296 + (-0.3) + 0.152 + 0.384 \\ &= 0.353 \text{ m of turpentine} \end{aligned}$$

or

$$p_1 - p_2 = 0.353(870)(9.81) = 3012 \text{ N/m}^2$$

$$\underline{p_1 - p_2 = 3.012 \text{ kPa}}$$

Figure 5.22. A pipeline for Example 5.6.

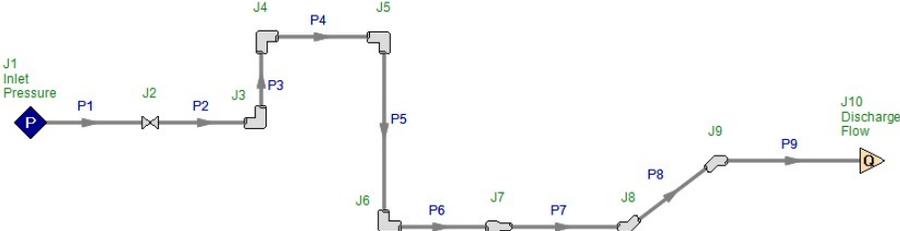


EXAMPLE 5.7

A water tank is fitted with a drain and outlet pipe as sketched in Figure 5.23. The system has 82 ft of cast iron pipe of 1½-nominal diameter. Determine the flow rate through the pipe. All fittings are threaded and regular.

View Verification Case 52 Model

[Verification Case 52](#)



Verification Case 53

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify53.fth

REFERENCE: William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Page 177-179, Example 5.7

FLUID: Water

ASSUMPTIONS: Water at 77F

RESULTS:

Parameter	Janna	AFT Fathom
Flow rate (ft ³ /sec)	0.0918	0.0905

DISCUSSION:

The various pipe lengths and intermediate junction elevations are not specified by Janna, so values were assigned to the AFT Fathom model such that the total length of 82 feet resulted, and the endpoint elevations were correct.

Results disagree slightly because Janna reads friction factors from a chart, which is less accurate than AFT Fathom's correlation based method.

[List of All Verification Models](#)

Verification Case 53 Problem Statement

Verification Case 53

William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Page 177-179, Example 5.7

Janna Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

The minor losses in the 12-pipe are from one gate valve and four standard elbows:

$$\Sigma K_{12} = 0.15 + 4(0.31) = 1.39$$

The minor losses in the 8-pipe area are from the contraction ($D_8^2/D_{12}^2 = 0.45$) and two 45° elbows:

$$\Sigma K_8 = 0.29 + 2(0.17) = 0.63$$

After rearranging the Bernoulli equation and substituting, we obtain

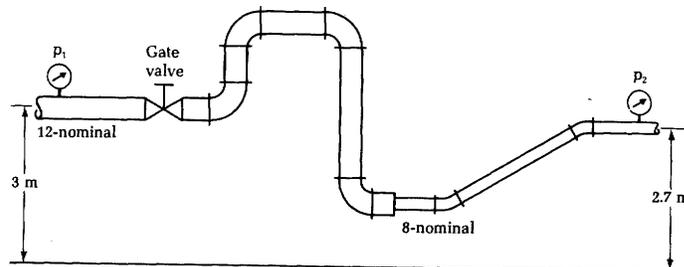
$$\begin{aligned} \frac{p_1 - p_2}{\rho} \frac{g_c}{g} &= \frac{(1.697)^2}{2(9.81)} - \frac{(0.762)^2}{2(9.81)} + (2.7 - 3) \\ &+ \left[\frac{0.018(60)}{0.289} + 1.39 \right] \frac{(0.762)^2}{2(9.81)} \\ &+ \left[\frac{0.0175(22)}{0.194} + 0.63 \right] \frac{(1.697)^2}{2(9.81)} \\ &= 0.1468 - 0.0296 + (-0.3) + 0.152 + 0.384 \\ &= 0.353 \text{ m of turpentine} \end{aligned}$$

or

$$p_1 - p_2 = 0.353(870)(9.81) = 3012 \text{ N/m}^2$$

$$p_1 - p_2 = 3.012 \text{ kPa}$$

Figure 5.22. A pipeline for Example 5.6.



EXAMPLE 5.7

A water tank is fitted with a drain and outlet pipe as sketched in Figure 5.23. The system has 82 ft of cast iron pipe of 1½-nominal diameter. Determine the flow rate through the pipe. All fittings are threaded and regular.

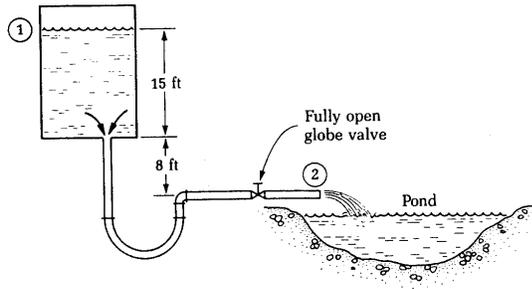


Figure 5.23. Piping system for Example 5.7.

SOLUTION

First write the Bernoulli equation from 1 to 2:

$$\frac{p_1 g_c}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{p_2 g_c}{\rho g} + \frac{V_2^2}{2g} + z_2 + \sum \frac{fL}{D} \frac{V^2}{2g} + \sum K \frac{V^2}{2g}$$

The figure shows that $p_1 = p_2$ and that if $z_2 = 0$ then $z_1 = 23$ ft. The reservoir surface velocity V_1 is negligible compared to V_2 . Because all the piping is of the same diameter, the equation becomes

$$23 = \frac{V_2^2}{2g} \left(1 + \frac{fL}{D} + \Sigma K \right)$$

The minor losses include the entrance, the return bend, the elbow and the globe valve. If we include also the loss due to the exit fitting, then the exit kinetic energy term in the above equation $V_2^2/2g$ should be zero. It is redundant to include the exit loss and a nonzero exit kinetic energy term. Thus we have

$$\Sigma K = 0.5 + 1.4 + 10.0 = 13.4 \quad (\text{Table 5.4})$$

$$D = 0.1342 \text{ ft} \quad \text{and} \quad A = 0.01414 \text{ ft}^2 \quad (\text{Appendix Table C-1})$$

$$\rho = 62.4 \text{ lbm/ft}^3 \quad \text{and} \quad \mu g_c = 6 \times 10^{-4} \text{ lbm/ft-s} \quad (\text{Appendix Table A-5})$$

$$\epsilon = 0.00085 \text{ ft} \quad (\text{Table 5-2})$$

By substitution, the Bernoulli equation becomes

$$23(2)(32.2) = V_2^2 \left[1 + \frac{f(82)}{0.1342} + 13.4 \right]$$

or

$$V_2 = \left(\frac{1481.2}{1 + \frac{f(82)}{0.1342} + 13.4} \right)^{1/2}$$

Because V_2 and f are unknown, a trial and error solution is required. As a first trial assume $f = 0.03$; then

$$\begin{aligned} V_2 &= 6.72 \text{ ft/s} \\ \text{Re} &= \frac{\rho V D}{\mu g_c} = \frac{62.4(6.72)(0.1342)}{6 \times 10^{-4}} = 9.38 \times 10^4 \\ \frac{\epsilon}{D} &= \frac{0.00085}{0.1342} = 0.0063 \end{aligned} \quad \left. \begin{array}{l} \text{Moody diagram:} \\ f = 0.0335 \end{array} \right\}$$

As a second trial assume $f = 0.0335$; then

$$\begin{aligned} V_2 &= 6.51 \text{ ft/s} \\ \text{Re} &= 9.1 \times 10^4 \\ \frac{\epsilon}{D} &= 0.0063 \end{aligned} \quad \left. \begin{array}{l} \text{Moody diagram} \\ f = 0.034 \text{ (close enough)} \end{array} \right\}$$

Using $f = 0.034$, $V_2 = 6.49 \text{ ft/s}$, and the continuity equation, we find

$$Q = A_2 V_2 = 0.01414(6.49)$$

$$Q = 0.0918 \text{ ft}^3/\text{s} = 5.51 \text{ ft}^3/\text{min}$$

EXAMPLE 5.8

Methyl alcohol is used in a processing plant where a flow rate of $0.8 \text{ m}^3/\text{min}$ of the liquid must be supplied. The available liquid pump can supply this flow rate only if the pressure drop in the supply line is less than 15 m head of water. The pipeline is made up of 50 m of soldered drawn copper tubing and follows the path shown in Figure 5.24. Determine the minimum size of tubing required.

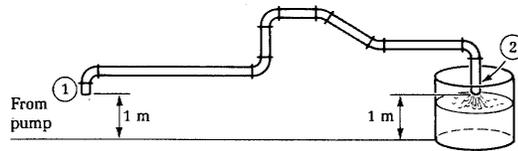


Figure 5.24. Sketch for Example 5.8.

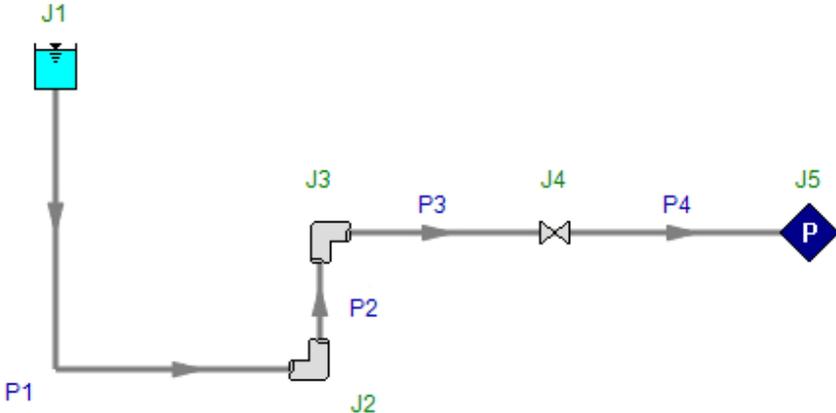
SOLUTION

Write the Bernoulli equation from 1 to 2 as

$$\frac{P_1 g_c}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{P_2 g_c}{\rho g} + \frac{V_2^2}{2g} + z_2 + \sum \frac{fL}{D} \frac{V^2}{2g} + \sum K \frac{V^2}{2g}$$

View Verification Case 53 Model

[Verification Case 53](#)



Verification Case 54

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify54.fth

REFERENCE: William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Page 183-186, Example 5.9

FLUID: Water

ASSUMPTIONS: Water at 77 deg. F

RESULTS:

Parameter	Janna	AFT Fathom
Flow rate pipe 2 (ft ³ /sec)	0.19	0.1947
Flow rate pipe 3 (ft ³ /sec)	0.11	0.1053

DISCUSSION:

The problem statement does not have any pressure information, because it is not required if the only purpose is to calculate flow rates. However, AFT Fathom requires at least one pressure boundary because AFT Fathom offers pressure information in the output for all systems. Therefore, a pressure of 1 atm was assumed for the common junction at the exit of the system.

An "out of balance" warning may appear in the AFT Fathom Output. The amount that is out of balance is very small and therefore the warning can be neglected.

[List of All Verification Models](#)

Verification Case 54 Problem Statement

Verification Case 54

William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Page 183-186, Example 5.9

Janna Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

(A_2 or A_3) is the same. Usually the diameters of all the pipes are known; the problem involves finding Q_2 and Q_3 . The solution technique requires use of the Moody diagram and trial and error. The technique is best illustrated by example.

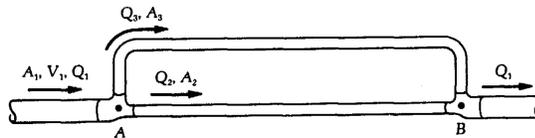


Figure 5.26. Pipes in parallel.

EXAMPLE 5.9

For the piping system of Figure 5.26, the distance from A to B is 4500 ft and the main line is made of schedule 40, 10-nominal wrought iron pipe. The attached loop is schedule 40, 8-nominal wrought iron pipe. The flow Q_1 is 0.3 ft³/s of water. Determine the flow rate in both branches.

SOLUTION

Minor losses are usually neglected in problems of this type. From the continuity equation,

$$Q_1 = Q_2 + Q_3 = 0.3 \text{ ft}^3/\text{s}$$

Also $p_A - p_B$ along Q_2 must be equal to $p_A - p_B$ along Q_3 . Thus

$$(p_A - p_B)_2 = (p_A - p_B)_3$$

or

$$\frac{f_2 L_2 V_2^2}{D_2 2g} = \frac{f_3 L_3 V_3^2}{D_3 2g}$$

With $L_2 = L_3$, the equation reduces to

$$\frac{f_2 V_2^2}{D_2} = \frac{f_3 V_3^2}{D_3}$$

In terms of volume flow rate,

$$\frac{f_2 16Q_2^2}{D_2 \pi^2 D_2^4} = \frac{f_3 16Q_3^2}{D_3 \pi^2 D_3^4}$$

or

$$\frac{f_2 Q_2^2}{D_2^5} = \frac{f_3 Q_3^2}{D_3^5}$$

From Appendix Table C-1,

$$\begin{aligned} D_2 &= 0.8350 \text{ ft} & \text{and} & & A_2 &= 0.5476 \text{ ft}^2 \\ D_3 &= 0.6651 \text{ ft} & \text{and} & & A_3 &= 0.3474 \text{ ft}^2 \end{aligned}$$

By substitution, the equation now becomes

$$2.46 f_2 Q_2^2 = 7.68 f_3 Q_3^2$$

or

$$Q_2 = Q_3 \sqrt{\frac{3.119 f_3}{f_2}}$$

From Table 5.2, $\epsilon = 0.00015$; hence

$$\frac{\epsilon}{D_2} = 0.000180$$

$$\frac{\epsilon}{D_3} = 0.000226$$

From Appendix Table A-5, $\rho = 62.4 \text{ lbm/ft}^3$ and $\mu_{g_c} = 6 \times 10^{-4} \text{ lbm/ft}\cdot\text{s}$.
The Reynolds number is

$$\text{Re} = \frac{\rho V D}{\mu_{g_c}} = \frac{\rho D}{\mu_{g_c}} \frac{4Q}{\pi D^2} = \frac{4\rho Q}{\pi D \mu_{g_c}}$$

For each line,

$$\text{Re}_2 = \frac{4(62.4)Q_2}{\pi(0.8350)(6 \times 10^{-4})} = 1.586 \times 10^5 Q_2$$

$$\text{Re}_3 = 1.991 \times 10^5 Q_3$$

From the calculations above, our working equations are

$$Q_2 + Q_3 = Q = 0.3 \text{ ft}^3/\text{s}$$

$$\text{Re}_2 = 1.586 \times 10^5 Q_2 \quad \epsilon/D_2 = 0.000180$$

$$\text{Re}_3 = 1.991 \times 10^5 Q_3 \quad \epsilon/D_3 = 0.000226$$

$$Q_2 = Q_3 \sqrt{\frac{3.119 f_3}{f_2}}$$

Because $D_2 > D_3$, we expect that $Q_2 > Q_3$.

As a first trial assume $Q_2 = \frac{1}{2}Q = 0.15 \text{ ft}^3/\text{s}$; then

$$Q_3 = 0.30 - 0.15 = 0.15$$

$$\text{Re}_2 = 2.37 \times 10^4$$

$$\text{Re}_3 = 2.99 \times 10^4$$

From the Moody diagram,

$$f_2 = 0.0255$$

$$f_3 = 0.0241$$

and by substitution into the working equation we get

$$Q_2 = 0.258 \text{ ft}^3/\text{s}$$

If this value is used for the second trial, the next iteration gives $Q_2 = 0.0898$. The third iteration gives $Q_2 = 0.33$, which shows that successive trials lead to divergence. A more successful method involves taking as a second assumed value the average of the initially assumed Q_2 and the calculated Q_2 . For the second iteration, therefore, assume

$$Q_2 = \frac{0.15 + 0.26}{2} = 0.20$$

then

$$Q_3 = 0.1$$

$$\text{Re}_2 = 3.17 \times 10^4$$

$$\text{Re}_3 = 1.99 \times 10^4$$

$$f_2 = 0.024$$

$$f_3 = 0.0265$$

and, by equation,

$$Q_2 = 0.18$$

For the next trial assume

$$Q_2 = \frac{0.18 + 0.20}{2} = 0.19$$

then

$$Q_3 = 0.11$$

$$\text{Re}_2 = 3.013 \times 10^4$$

$$\text{Re}_3 = 2.19 \times 10^4$$

$$f_2 = 0.024$$

$$f_3 = 0.026$$

and, by equation,

$$Q_2 = 0.20$$

which is close enough to the assumed value. Thus the solution is

$$\underline{Q_2 = 0.19 \text{ ft}^3/\text{s} \quad \text{and} \quad Q_3 = 0.11 \text{ ft}^3/\text{s}}$$

All pipe friction problems in this chapter have been solved by using the Moody diagram. Although reference to a figure is highly impractical if one is using a computer for the calculations, the computer is well suited for these problems, especially the trial and error type. Thus an equation fit of the Moody diagram is helpful. Among the many equations that have been written are the following:

Hagen-Poiseuille equation:

$$f = \frac{64}{\text{Re}} \quad (\text{laminar flow, circular pipe}) \quad (5.35)$$

Prandtl equation:

$$\frac{1}{\sqrt{f}} = 2.0 \log (\text{Re} \sqrt{f}) - 0.8 \quad (3\,000 < \text{Re} < 3.4 \times 10^6) \quad (5.36)$$

Von Karman equation:

$$\frac{1}{\sqrt{f}} = 2.0 \log \left(\frac{D}{\epsilon} \right) + 1.74 \quad \frac{D}{\epsilon} \frac{1}{\text{Re} \sqrt{f}} > 0.01 \quad (5.37)$$

Colebrook equation:

$$\frac{1}{\sqrt{f}} = -2.0 \log \left(\frac{\epsilon}{3.7065D} + \frac{2.5226}{\text{Re} \sqrt{f}} \right) \quad \frac{D}{\epsilon} \frac{1}{\text{Re} \sqrt{f}} < 0.005 \quad (5.38)$$

Except for the laminar-flow expression, all these equations have \sqrt{f} appearing on both sides or in the condition. This difficulty is overcome by the Chen equation, which is good for any ϵ/D and Re :

$$\frac{1}{\sqrt{f}} = -2.0 \log \left\{ \frac{\epsilon}{3.7065D} - \frac{5.0452}{\text{Re}} \log \left[\frac{1}{2.8257} \left(\frac{\epsilon}{D} \right)^{1.1098} + \frac{5.8506}{\text{Re}^{0.0981}} \right] \right\} \quad (5.39)$$

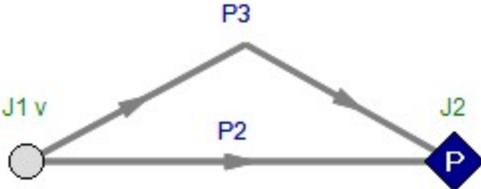
If ϵ/D and Re are known then f can be determined, just as with the Moody diagram.

5.10 / Pumps and Piping Systems

To make fluid flow from one point to another in a closed conduit, a driving force is required. Where elevation differences exist this force may be grav-

View Verification Case 54 Model

[Verification Case 54](#)



Verification Case 55

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify55.fth

REFERENCE: William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Page 189-190, Example 5.10

FLUID: Water

ASSUMPTIONS: Water at 75 deg. F

RESULTS:

Parameter	Janna	AFT Fathom
Pump ideal power (hp)	0.737	0.734

DISCUSSION:

The losses due to elbows are included as a fitting and loss value in the pipes. The ideal power usage is given the Output window Pump Summary.

[List of All Verification Models](#)

Verification Case 55 Problem Statement

Verification Case 55

William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Page 189-190, Example 5.10

Janna Title Page

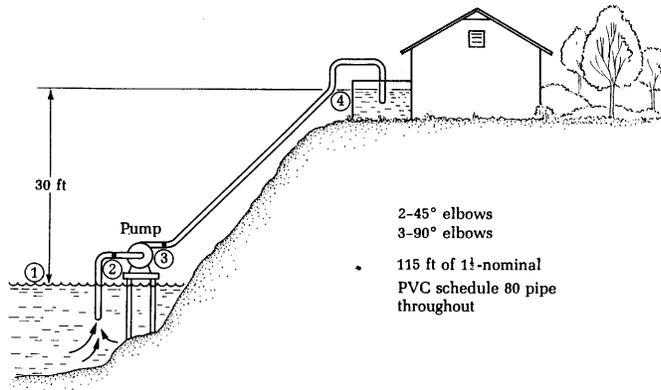
Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

which is converted to pressure head after the liquid leaves the impeller. Pumping power requirements are determined with the energy equation as illustrated by the following example.

EXAMPLE 5.10

A house is located near a freshwater lake. The homeowner decides to install a pump near the lake to deliver 50 gpm of water to a tank adjacent to the house. The water can then be used for lavatory facilities or sprinkling the lawn. For the system sketched in Figure 5.30, determine the pump power required.

Figure 5.30. Pumping system for Example 5.10.



SOLUTION

The following equations can be written for the system:

$$\frac{p_1 g_c}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{p_2 g_c}{\rho g} + \frac{V_2^2}{2g} + z_2 + \frac{V^2}{2g} \left(\frac{fL}{D} + \Sigma K \right)$$

$$\frac{p_2 g_c}{\rho g} + \frac{V_2^2}{2g} + z_2 = \frac{p_3 g_c}{\rho g} + \frac{V_3^2}{2g} + z_3 + \frac{1}{m} \frac{dW}{dt} \frac{g_c}{g}$$

$$\frac{p_3 g_c}{\rho g} + \frac{V_3^2}{2g} + z_3 = \frac{p_4 g_c}{\rho g} + \frac{V_4^2}{2g} + z_4 + \frac{V^2}{2g} \left(\frac{fL}{D} + \Sigma K \right)$$

These equations can be added to obtain a single equation for the entire system:

$$\frac{p_1 g_c}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{p_4 g_c}{\rho g} + \frac{V_4^2}{2g} + z_4 + \sum \frac{fL}{D} \frac{V_4^2}{2g} + \sum K \frac{V_4^2}{2g} + \frac{1}{\dot{m}} \frac{dW}{dt} \frac{g_c}{g}$$

Thus only inlet and outlet conditions must be known. At sections 1 and 4 (both are reservoir surfaces), velocity is negligible. With section 1 as the reference,

$$-\frac{1}{\dot{m}} \frac{dW}{dt} = \frac{z_4 g}{g_c} + \frac{V_4^2}{2g_c} \left(1 + \frac{fL}{D} + \Sigma K \right)$$

The volume flow required is

$$Q = 50 \text{ gpm} \times 0.002228 = 0.111 \text{ ft}^3/\text{s}$$

For water, $\rho = 62.4 \text{ lbm/ft}^3$ and, from Appendix Table A-4, $\mu_{g_c} = 6.122 \times 10^{-4}$; so

$$\dot{m} = \rho Q = 6.926 \text{ lbm/s}$$

From Appendix Table C-1 for $1\frac{1}{2}$ -nominal schedule 80 pipe, we find

$$D = 0.125 \text{ ft} \quad \text{and} \quad A = 0.01227 \text{ ft}^2$$

The velocity is

$$V = \frac{Q}{A} = \frac{0.111}{0.01227} = 9.05 \text{ ft/s}$$

with which we obtain

$$\text{Re} = \frac{62.4(9.05)(0.125)}{6.122 \times 10^{-4}} = 1.15 \times 10^5$$

It is permissible to use the "smooth pipe" curve on the Moody diagram for PVC pipe. At $\text{Re} = 1.15 \times 10^5$, therefore, $f = 0.018$. From Table 5.4, $K = 1.4$ for 90° elbows, $K = 0.35$ for 45° elbows and 1.0 for an exit fitting. By substitution into the equation, we obtain

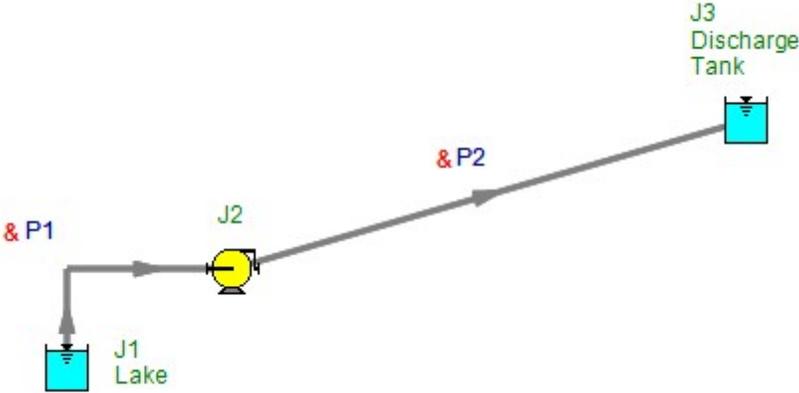
$$\begin{aligned} -\frac{dW}{dt} &= 6.926 \left\{ 30 + \frac{(9.05)^2}{2(32.2)} \left[1 + \frac{0.018(115)}{0.125} + 2(0.35) + 3(1.4) \right] \right\} \\ &= 6.926(58.56) = 405.6 \text{ ft-lbf/s} \end{aligned}$$

By definition, 1 hp = 550 ft-lbf/s; therefore

$$\frac{dW}{dt} = -0.737 \text{ hp}$$

View Verification Case 55 Model

[Verification Case 55](#)



Verification Case 56

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify56.fth

REFERENCE: Nicholas P. Chohey, Handbook of Chemical Engineering Calculations, 1994, McGraw-Hill, Page 6-4, Example 6-3

FLUID: Fuel Oil at 300 deg. F

ASSUMPTIONS: No elevation changes. Inlet pressure is 100 psia.

RESULTS:

Parameter	Chohey	AFT Fathom
Pressure drop (psid)	1.17	1.172

DISCUSSION:

No absolute pressure information was provided. It was assumed the inlet pressure was 100 psia. This does not affect the answer.

The flow in the pipe is laminar.

[List of All Verification Models](#)

Verification Case 56 Problem Statement

[Verification Case 56](#)

Nicholas P. Chohey, Handbook of Chemical Engineering Calculations, 1994, McGraw-Hill, Page 6-4, Example 6-3

[Chohey Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

6-4 FLOW OF FLUIDS AND SOLIDS

6-3 Pressure Loss in Piping with Laminar Flow

Fuel oil at 300°F (422 K) and having a specific gravity of 0.850 is pumped through a 30,000-ft-long 24-in pipe at the rate of 500 gal/min (0.032 m³/s). What is the pressure loss if the viscosity of the oil is 75 cP?

Calculation Procedure:

1. Determine the type of flow that exists.

Flow is laminar (also termed viscous) if the Reynolds number Re for the liquid in the pipe is less than about 2000. Turbulent flow exists if the Reynolds number is greater than about 4000. Between these values is a zone in which either condition may exist, depending on the roughness of the pipe wall, entrance conditions, and other factors. Avoid sizing a pipe for flow in this critical zone because excessive pressure drops result without a corresponding increase in the pipe discharge.

Compute the Reynolds number from $Re = 3.162G/kd$, where G = flow rate, gal/min; k = kinematic viscosity of liquid, Cst = viscosity z cP/specific gravity of the liquid S ; d = inside diameter of pipe, in. From a table of pipe properties, $d = 22.626$ in. Also, $k = z/S = 75/0.85 = 88.2$ Cst. Then, $Re = 3162(500)/[88.2(22.626)] = 792$. Since $Re < 2000$, laminar flow exists in this pipe.

2. Compute the pressure loss using the Poiseuille formula.

The Poiseuille formula gives the pressure drop ρ_d lb/in² = $2.73(10^{-4})luG/d^4$, where l = total length of pipe, including equivalent length of fitting, ft; u = absolute viscosity of liquid, cP; G = flow rate, gal/min; d = inside diameter of pipe, in. For this pipe, $\rho_d = 2.73(10^{-4})(30,000)(75)(500)/262,078 = 1.17$ lb/in² (8.07 kPa).

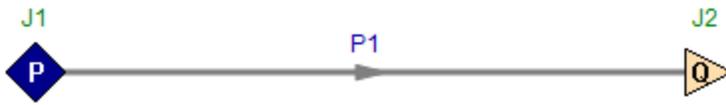
Related Calculations: Use this procedure for any pipe in which there is laminar flow of liquid. Table 6-1 gives a quick summary of various ways in which the Reynolds num-

TABLE 6-1 Reynolds Number

Reynolds number Re	Numerator			Denominator		
	Coefficient	First symbol	Second symbol	Third symbol	Fourth symbol	Fifth symbol
$Dv\rho/\mu$	—	ft	ft/s	lb/ft ³	lb mass/ft·s	—
$124dv\rho/z$	124	in	ft/s	lb/ft ³	cP	—
$50.7G\rho/dz$	50.7	gal/min	lb/ft ³	—	in	cP
$6.32W/dz$	6.32	lb/h	—	—	in	cP
$35.5B\rho/dz$	35.5	bbl/h	lb/ft ³	—	in	cP
$7,742dv/k$	7,742	in	ft/s	—	—	cP
$3,162G/dk$	3,162	gal/min	—	—	in	cP
$2,214B/dk$	2,214	bbl/h	—	—	in	cP
$22,735q\rho/dz$	22,735	ft ³ /s	lb/ft ³	—	in	cP
$378.9Q\rho/dz$	378.9	ft ³ /min	lb/ft ³	—	in	cP

View Verification Case 56 Model

[Verification Case 56](#)



Verification Case 57

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify57.fth

REFERENCE: Nicholas P. Chohey, Handbook of Chemical Engineering Calculations, 1994, McGraw-Hill, Page 6-5, 11, Example 6-4

FLUID: Kerosene at 65 deg. F

ASSUMPTIONS: No elevation changes. Inlet pressure is 100 psia.

RESULTS:

Parameter	Chohey	AFT Fathom
Pressure drop (psid)	17.3	17.24

DISCUSSION:

The kinematic viscosity of 2.95 cSt was converted to dynamic viscosity, obtaining 0.00161 lbm/ft-s.

No absolute pressure information was provided. It was assumed the inlet pressure was 100 psia. This does not affect the answer.

[List of All Verification Models](#)

Verification Case 57 Problem Statement

[Verification Case 57](#)

Nicholas P. Chopey, Handbook of Chemical Engineering Calculations, 1994, McGraw-Hill, Page 6-5, 11, Example 6-4

[Chopey Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

DETERMINING THE PRESSURE LOSS IN PIPES 6-5

ber can be expressed. The symbols in Table 6-1, in the order of their appearance, are D = inside diameter of pipe, ft; v = liquid velocity, ft/s; ρ = liquid density, lb/ft³; μ = absolute viscosity of liquid, lb mass/ft·s; d = inside diameter of pipe, in. From a table of pipe properties, $d = 22.626$ in. Also, $k = z/S$ liquid flow rate, lb/h; B = liquid flow rate, bbl/h; k = kinematic viscosity of the liquid, cSt; q = liquid flow rate, ft³/s; Q = liquid flow rate, ft³/min. Use Table 6-1 to find the Reynolds number for any liquid flowing through a pipe.

6-4 Determining the Pressure Loss in Pipes

What is the pressure drop in a 5000-ft-long 6-in oil pipe conveying 500 bbl/h (0.022 m³/s) kerosene having a specific gravity of 0.813 at 65°F, which is the temperature of the liquid in the pipe? The pipe is schedule 40 steel.

Calculation Procedure:

1. Determine the kinematic viscosity of the oil.

Use Fig. 6-1 and Table 6-2 or the Hydraulic Institute—*Pipe Friction Manual* kinematic viscosity and Reynolds number chart to determine the kinematic viscosity of the liquid. Enter Table 6-2 at kerosene and find the coordinates as $X = 10.2$, $Y = 16.9$. Using these coordinates, enter Fig. 6-1 and find the absolute viscosity of kerosene at 65°F as 2.4 cP. Using the method of Example 6-2, the kinematic viscosity, in cSt, equals absolute viscosity, cP/specific gravity of the liquid = $2.4/0.813 = 2.95$ cSt. This value agrees closely with that given in the *Pipe Friction Manual*.

2. Determine the Reynolds number of the liquid.

The Reynolds number can be found from the *Pipe Friction Manual* chart mentioned in step 1 or computed from $Re = 2214 B/dk = 2214(500)/[(6.065)(2.95)] = 61,900$.

To use the *Pipe Friction Manual* chart, compute the velocity of the liquid in the pipe by converting the flow rate to cubic feet per second. Since there are 42 gal/bbl and 1 gal = 0.13368 ft³, 1 bbl = $(42)(0.13368) = 5.6$ ft³. With a flow rate of 500 bbl/h, the equivalent flow in ft³ = $(500)(5.6) = 2800$ ft³/h, or $2800/3600$ s/h = 0.778 ft³/s. Since 6-in schedule 40 pipe has a cross-sectional area of 0.2006 ft² internally, the liquid velocity, in ft/s, equals $0.778/0.2006 = 3.88$ ft/s. Then, the product (velocity, ft/s)(internal diameter, in) = $(3.88)(6.065) = 23.75$. In the *Pipe Friction Manual*, project horizontally from the kerosene specific-gravity curve to the vd product of 23.75 and read the Reynolds number as 61,900, as before. In general, the Reynolds number can be found faster by computing it using the appropriate relation given in Table 6-1, unless the flow velocity is already known.

3. Determine the friction factor of this pipe.

Enter Fig. 6-2 at the Reynolds number value of 61,900 and project to the curve 4 as indicated by Table 6-3. Read the friction factor as 0.0212 at the left. Alternatively, the *Pipe Friction Manual* friction-factor chart could be used, if desired.

VISCOSITIES

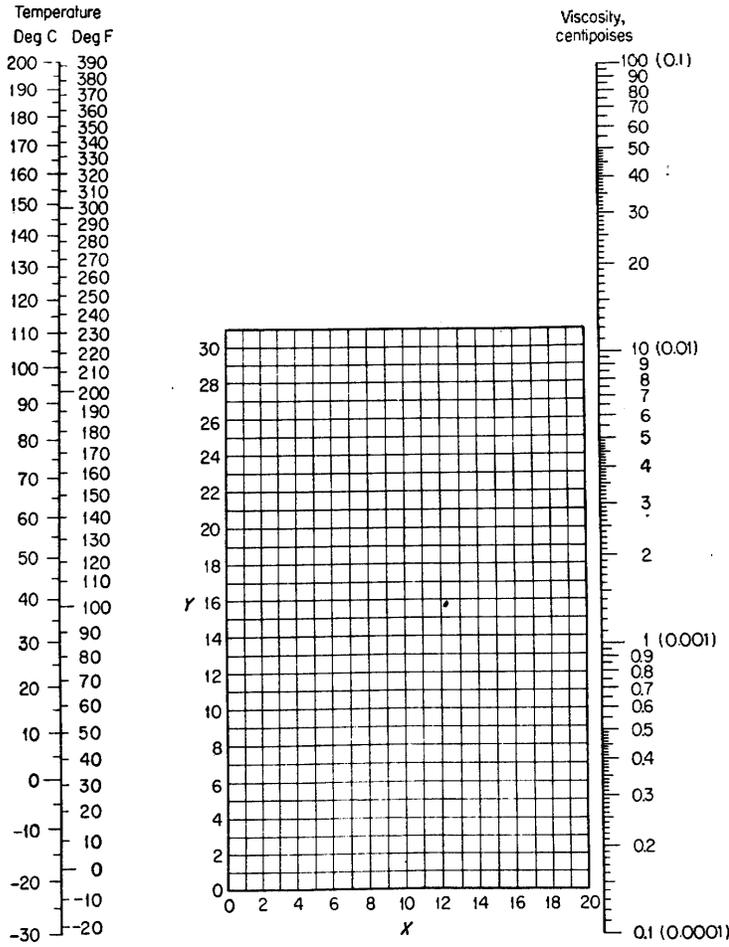


FIG. 6-1 Viscosities of liquids at 1 atm (101.3 kPa). For coordinates, see Table 6-2.

Verification Case 57 Problem Statement

TABLE 6-2 Viscosities of Liquids (coordinates for use with Fig. 6-1)

No.	Liquid	X	Y	No.	Liquid	X	Y
1	Acetaldehyde	15.2	4.8	56	Freon-22	17.2	4.7
	Acetic acid:			57	Freon-13	12.5	11.4
2	100%	12.1	14.2		Glycerol:		
3	70%	9.5	17.0	58	100%	2.0	30.0
4	Acetic anhydride	12.7	12.8	59	50%	6.9	19.6
	Acetone:			60	Heptene	14.1	8.4
5	100%	14.5	7.2	61	Hexane	14.7	7.0
6	35%	7.9	15.0	62	Hydrochloric acid, 31.5%	13.0	16.6
7	Allyl alcohol	10.2	14.3	63	Isobutyl alcohol	7.1	18.0
	Ammonia:			64	Isobutyric acid	12.2	14.4
8	100%	12.6	2.0	65	Isopropyl alcohol	8.2	16.0
9	26%	10.1	13.9	66	Kerosene	10.2	16.9
10	Amyl acetate	11.8	12.5	67	Linseed oil, raw	7.5	27.2
11	Amyl alcohol	7.5	18.4	68	Mercury	18.4	16.4
12	Aniline	8.1	18.7		Methanol		
13	Anisole	12.3	13.5	69	100%	12.4	10.5
14	Arsenic trichloride	13.9	14.5	70	90%	12.3	11.8
15	Benzene	12.5	10.9	71	40%	7.8	15.5
	Brine:			72	Methyl acetate	14.2	8.2
16	CaCl ₂ , 25%	6.6	15.9	73	Methyl chloride	15.0	3.8
17	NaCl, 25%	10.2	16.6	74	Methyl ethyl ketone	13.9	8.6
18	Bromine	14.2	13.2	75	Naphthalene	7.9	18.1
19	Bromotoluene	20.0	15.9		Nitric acid:		
20	Butyl acetate	12.3	11.0	76	95%	12.8	13.8
21	Butyl alcohol	8.6	17.2	77	60%	10.8	17.0
22	Butyric acid	12.1	15.3	78	Nitrobenzene	10.6	16.2
23	Carbon dioxide	11.6	0.3	79	Nitrotoluene	11.0	17.0
24	Carbon disulfide	16.1	7.5	80	Octane	13.7	10.0
25	Carbon tetrachloride	12.7	13.1	81	Octyl alcohol	6.6	21.1
26	Chlorobenzene	12.3	12.4	82	Pentachloroethane	10.9	17.3
27	Chloroform	14.4	10.2	83	Pentane	14.9	5.2
28	Chlorosulfonic acid	11.2	18.1	84	Phenol	6.9	20.8
	Chlorotoluene:			85	Phosphorus tribromide	13.8	16.7
29	Ortho	13.0	13.3	86	Phosphorus trichloride	16.2	10.9
30	Meta	13.3	12.5	87	Propionic acid	12.8	13.8
31	Para	13.3	12.5	88	Propyl alcohol	9.1	16.5
32	Cresol, meta	2.5	20.8	89	Propyl bromide	14.5	9.6
33	Cyclohexanol	2.9	24.3	90	Propyl chloride	14.4	7.5
34	Dibromoethane	12.7	15.8	91	Propyl iodide	14.1	11.6
35	Dichloroethane	13.2	12.2	92	Sodium	16.4	13.9
36	Dichloromethane	14.6	8.9	93	Sodium hydroxide, 50%	3.3	25.8
37	Diethyl oxalate	11.0	16.4	94	Stannic chloride	13.5	12.8
38	Dimethyl oxalate	12.3	15.8	95	Sulfur dioxide	15.2	7.1
39	Diphenyl	12.0	18.3		Sulfuric acid:		
40	Dipropyl oxalate	10.3	17.7	96	110%	7.2	27.4
41	Ethyl acetate	13.7	9.1	97	98%	7.0	24.8
	Ethyl alcohol:			98	60%	10.2	21.3
42	100%	10.5	13.8	99	Sulfuryl chloride	15.2	12.4
43	95%	9.8	14.3	100	Tetrachloroethane	11.9	15.7
44	40%	6.5	16.6	101	Tetrachloroethylene	14.2	12.7
45	Ethyl benzene	13.2	11.5	102	Titanium tetrachloride	14.4	12.3
46	Ethyl bromide	14.5	8.1	103	Toluene	13.7	10.4
47	Ethyl chloride	14.8	6.0	104	Trichloroethylene	14.8	10.5
48	Ethyl ether	14.5	5.3	105	Turpentine	11.5	14.9
49	Ethyl formate			106	Vinyl acetate		
50	Ethyl iodide	14.7	10.3	107	Water	10.2	13.0
51	Ethylene glycol	6.0	23.6		Xylene:		
52	Formic acid	10.7	15.8	108	Ortho	13.5	12.1
53	Freon-11	14.4	9.0	109	Meta	13.9	10.6
54	Freon-12	16.8	5.6	110	Para	13.9	10.9
55	Freon-21	15.7	7.5				

8-9

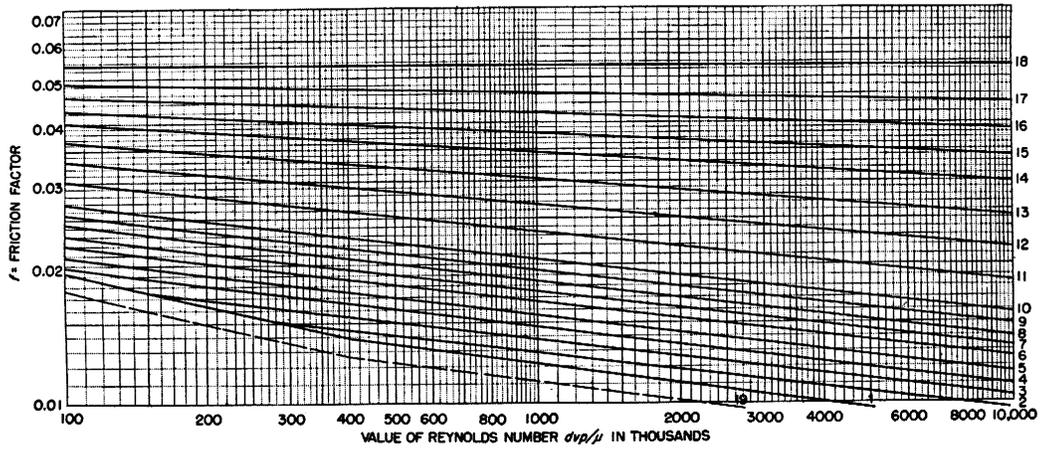
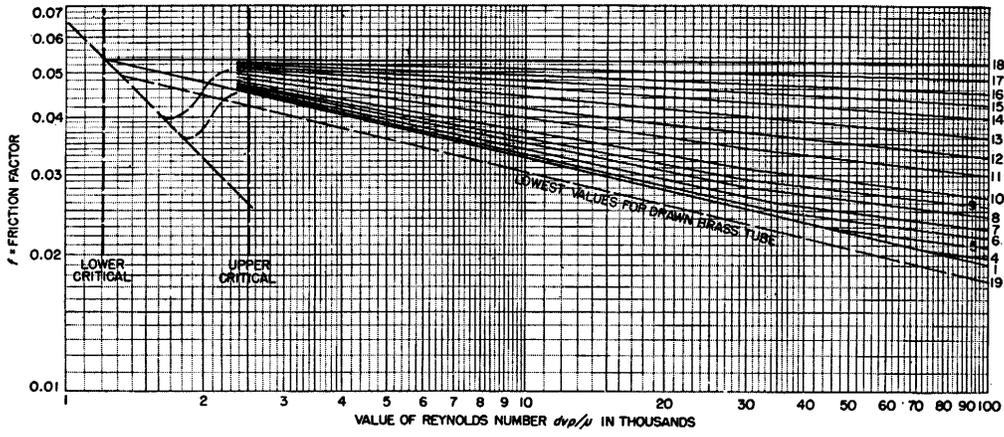


FIG. 6-2 Friction-factor curves. (Mechanical Engineering.)

6-9

Verification Case 57 Problem Statement

01-9

TABLE 6-3 Data for Fig. 6-2

Percent roughness	For value of <i>f</i> see curve	Diameter (actual of drawn tubing, nominal of standard-weight pipe)											
		Drawn tubing, brass, tin, lead, glass		Clean steel, wrought iron		Clean, galvanized		Best cast iron		Average cast iron		Heavy riveted, spiral riveted	
		in	mm	in	mm	in	mm	in	mm	in	mm	in	mm
0.2	1	0.35 up	8.89 up	72	1829	—	—	—	—	—	—	—	—
1.35	4	—	—	6-12	152-305	10-24	254-610	20-48	508-1219	42-96	1067-2438	84-204	2134-5182
2.1	5	—	—	4-5	102-127	6-8	152-203	12-16	305-406	24-36	610-914	48-72	1219-1829
3.0	6	—	—	2-3	51-76	305	76-127	5-10	127-254	10-20	254-508	20-42	508-1067
3.8	7	—	—	1½	38	2½	64	3-4	76-102	6-8	152-203	16-18	406-457
4.8	8	—	—	1-1½	25-32	1½-2	38-51	2-2½	51-64	4-5	102-127	10-14	254-356
6.0	9	—	—	¾	19	1¼	32	1½	38	3	76	8	203
7.2	10	—	—	½	13	1	25	1¼	32	—	—	5	127
10.5	11	—	—	¾	9.5	¾	19	1	35	—	—	4	102
14.5	12	—	—	½	6.4	½	13	—	—	—	—	3	76
24.0	14	0.125	3.18	—	—	¾	9.5	—	—	—	—	—	—
31.5	16	—	—	—	—	¾	6.4	—	—	—	—	—	—
37.5	18	0.0625	1.588	—	—	¾	3.2	—	—	—	—	—	—

EQUIVALENT LENGTH OF A COMPLEX-SERIES PIPELINE 6-11

4. Compute the pressure loss in the pipe.

Use the Fanning formula $p_d = 1.06(10^{-4})f\rho lB^2/d^5$. In this formula, ρ = density of the liquid, lb/ft³. For kerosene, $p = (\text{density of water, lb/ft}^3)(\text{specific gravity of the kerosene}) = (62.4)(0.813) = 50.6 \text{ lb/ft}^3$. Then, $p_d = 1.06(10^{-4})(0.0212)(50.6)(5000)(500)^2/8206 = 17.3 \text{ lb/in}^2 (119 \text{ kPa})$.

Related Calculations: The Fanning formula is popular with oil-pipe designers and can be stated in various ways: (1) with velocity v , in ft/s, $p_d = 1.29(10^{-3})f\rho v^2 l/d$; (2) with velocity V , in ft/min, $p_d = 3.6(10^{-7})f\rho V^2 l/d$; (3) with flow rate G , in gal/min, $p_d = 2.15(10^{-4})f\rho G^2/d^5$; (4) with the flow rate W , in lb/h, $p_d = 3.36(10^{-6})f l W^2/d^5 \rho$.

Use this procedure for any fluid—crude oil, kerosene, benzene, gasoline, naphtha, fuel oil, Bunker C, diesel oil toluene, etc. The tables and charts presented here and in the *Pipe Friction Manual* save computation time.

6-5 Equivalent Length of a Complex-Series Pipeline

Figure 6-3 shows a complex-series pipeline made up of four lengths of different size pipe. Determine the equivalent length of this pipe if each size of pipe has the same friction factor.

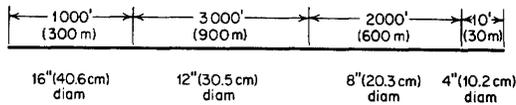


FIG. 6-3 Complex-series pipeline.

Calculation Procedure:

1. Select the pipe size for expressing the equivalent length.

The usual procedure when analyzing complex pipelines is to express the equivalent length in terms of the smallest, or next-to-smallest, diameter pipe. Choose the 8-in size as being suitable for expressing the equivalent length.

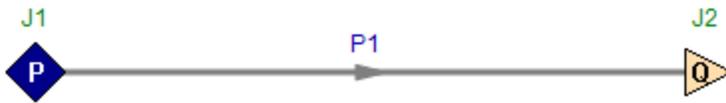
2. Find the equivalent length of each pipe.

For any complex-series pipeline having equal friction factors in all the pipes, L_e = equivalent length, ft, of a section of constant diameter = (actual length of section, ft) (inside diameter, in, of pipe used to express the equivalent length/inside diameter, in, of section under consideration)⁵.

For the 16-in pipe, $L_e = (1000)(7.981/15.000)^5 = 42.6 \text{ ft}$. The 12-in pipe is next; for it, $L_e = (3000)(7.981/12.00)^5 = 390 \text{ ft}$. For the 8-in pipe, the equivalent length = actual length = 2000 ft. For the 4-in pipe, $L_e = (10)(7.981/4.026)^5 = 306 \text{ ft}$. Then, the total equivalent length of 8-in pipe = sum of the equivalent lengths = 42.6 + 390 + 2000 + 306 = 2738.6 ft, or rounding off, 2740 ft of 8-in pipe (835 m of 0.2-m pipe) will have a frictional resistance equal to the complex-series pipeline shown in Fig. 6-3. To compute the actual frictional resistance, use the methods given in previous Calculation Procedures.

View Verification Case 57 Model

[Verification Case 57](#)



Verification Case 58

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify58.fth

REFERENCE: Nicholas P. Chopey, Handbook of Chemical Engineering Calculations, 1994, McGraw-Hill, Page 6-41, 46, Example 6-21

FLUID: Water

ASSUMPTIONS: Assume water at 70 deg. F.

RESULTS:

Parameter	Chopey	AFT Fathom
Head rise case A (feet)	125	124.4
Head rise case B (feet)	125	124.4
Head rise case C (feet)	125	124.4
Power usage case A (hp)	90.2	89.8
Power usage case B (hp)	90.2	89.8
Power usage case C (hp)	90.2	89.8

DISCUSSION:

The hydraulic loss data used by Chopey is based on equivalent length. All equivalent lengths were converted to K values and included as a fitting and loss value in the pipe or as custom entrance/exit loss factors at the tanks.

[List of All Verification Models](#)

Verification Case 58 Problem Statement

[Verification Case 58](#)

Nicholas P. Chohey, Handbook of Chemical Engineering Calculations, 1994, McGraw-Hill, Page 6-41, 46, Example 6-21

[Chohey Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

TOTAL HEAD ON A PUMP HANDLING VAPOR-FREE LIQUID 6-41

TABLE 6-21 Suction Specific-Speed Ratings

Single-suction pump	Double-suction pump	Rating
Above 11,000	Above 14,000	Excellent
9,000–11,000	11,000–14,000	Good
7,000–9,000	9,000–11,000	Average
5,000–7,000	7,000–9,000	Poor
Below 5,000	Below 7,000	Very poor

Source: Peerless Pump Division, FMC Corporation.

Related Calculations: Use this procedure for any type of centrifugal pump handling water for plant services, cooling, process, fire protection, and similar requirements. This procedure is the work of R. P. Horwitz, Hydrodynamics Division, Peerless Pump, FMC Corporation, as reported in *Power* magazine.

6-21 Total Head on a Pump Handling Vapor-Free Liquid

Sketch three typical pump piping arrangements with static suction lift and submerged, free, and varying discharge head. Prepare similar sketches for the same pump with static suction head. Label the various heads. Compute the total head on each pump if the elevations are as shown in Fig. 6-13 and the pump discharges a maximum of 2000 gal/min (0.126 m³/s) of water through 8-in schedule 40 pipe. What horsepower is required to drive the pump? A swing check valve is used on the pump suction line and a gate valve on the discharge line.

Calculation Procedure:

1. Sketch the possible piping arrangements.

Figure 6-13 shows the six possible piping arrangements for the stated conditions of the installation. Label the total static head—i.e., the vertical distance from the surface of the source of the liquid supply to the free surface of the liquid in the discharge receiver, or to the point of free discharge from the discharge pipe. When both the suction and discharge surfaces are open to the atmosphere, the total static head equals the vertical difference in elevation. Use the free-surface elevations that cause the maximum suction lift and discharge head—i.e., the lowest possible level in the supply tank and the highest possible level in the discharge tank or pipe. When the supply source is below the pump centerline, the vertical distance is called the “static suction lift.” With the supply above the pump centerline, the vertical distance is called “static suction head.” With variable static suction head, use the lowest liquid level in the supply tank when computing total static head. Label the diagrams as shown in Fig. 6-13.

2. Compute the total static head on the pump.

The total static head, in feet, is $H_s =$ static suction lift, in feet, $h_{sl} +$ static discharge head, in feet, h_{sd} , where the pump has a suction lift, s in Fig. 6-13a, b, and c. In these

6-42 FLOW OF FLUIDS AND SOLIDS

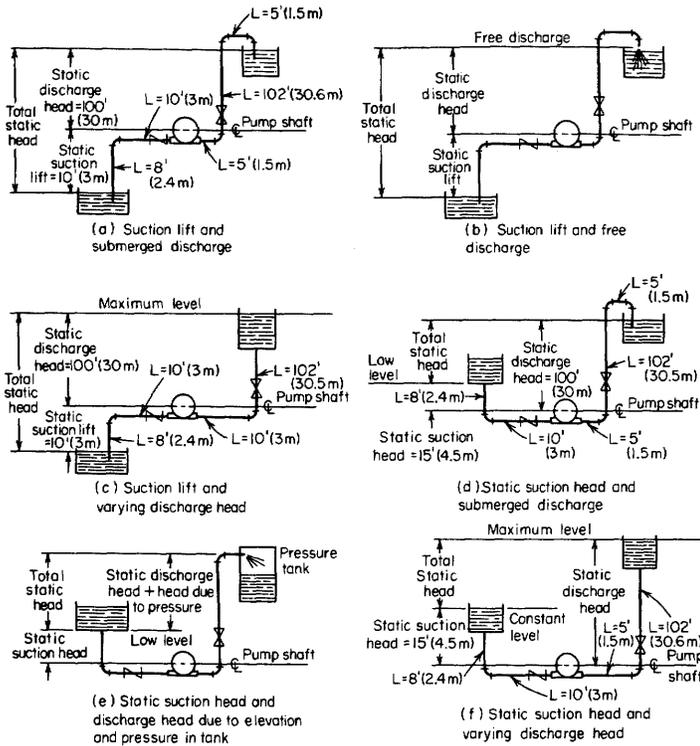


FIG. 6-13 Typical pump suction and discharge piping arrangements.

installations, $H_u = 10 + 100 = 110$ ft. Note that the static discharge head is computed between the pump centerline and the water level with an underwater discharge (Fig. 6-13a), to the pipe outlet with a free discharge (Fig. 6-13b), and to the maximum water level in the discharge tank (Fig. 6-13c). When a pump is discharging into a closed compression tank, the total discharge head equals the static discharge head plus the head equivalent, in feet of liquid, of the internal pressure in the tank, or $2.31 \times$ tank pressure, in lb/in^2 .

Where the pump has a static suction head, as in Fig. 6-13d, e, and f, the total static head, in feet, is $H_u = h_d - \text{static suction head}$, in feet, h_s . In these installations, $H_u = 100 - 15 = 85$ ft.

The total static head, as computed above, refers to the head on the pump without liquid flow. To determine the total head on the pump, the friction losses in the piping system during liquid flow must also be determined.

TOTAL HEAD ON A PUMP HANDLING VAPOR-FREE LIQUID 6-43

3. Compute the piping friction losses.

Mark the length of each piece of straight pipe on the piping drawing. Thus, in Fig. 6-13a, the total length of straight pipe L_s , in feet, is $8 + 10 + 5 + 102 + 5 = 130$ ft, starting at the suction tank and adding each length until the discharge tank is reached. To the total length of straight pipe must be added the equivalent length of the pipe fittings. In Fig. 6-13a there are four long-radius elbows, one swing check valve, and one globe valve. In addition, there is a minor head loss at the pipe inlet and at the pipe outlet.

The equivalent length of one 8-in-long-radius elbow is 14 ft of pipe, from Table 6-22. Since the pipe contains four elbows, the total equivalent length is $4(14) = 56$ ft of straight pipe. The open gate valve has an equivalent resistance of 4.5 ft, and the open swing check valve has an equivalent resistance of 53 ft.

The entrance loss h_e , in feet, assuming a basket-type strainer is used at the suction-pipe inlet, is $Kv^2/(2g)$, where $K =$ a constant from Fig. 6-14; $v =$ liquid velocity, in ft/s; and $g = 32.2$ ft/s². The exit loss occurs when the liquid passes through a sudden enlargement, as from a pipe to a tank. Where the area of the tank is large, causing a final velocity that is zero, $h_{ex} = v^2/2g$.

The velocity v , in feet per second, in a pipe is (gal/min)/(2.448d²). For this pipe, $v = 2000/[2.448(7.98)^2] = 12.82$ ft/s. Then, $h_e = 0.74(12.82)^2/[2(32.2)] = 1.89$ ft, and $h_{ex} = (12.82)^2/[2(32.2)] = 2.56$ ft (0.78 m). Hence the total length of the piping system in Fig. 6-13a is $130 + 56 + 4.5 + 53 + 1.89 + 2.56 = 247.95$ ft (75.6 m), say 248 ft (75.6 m).

Use a suitable head-loss equation, or Table 6-23, to compute the head loss for the pipe and fittings. Enter Table 6-23 at an 8-in (203.2-mm) pipe size and project horizontally across to 2000 gal/min (126.2 L/s) and read the head loss as 5.86 ft of water per 100 ft (1.8 m/30.5 m) of pipe.

The total length of pipe and fittings computed above is ²⁴⁸288 ft (87.8 m). Then, total friction-head loss with a 2000-gal/min (126.2 L/s) flow is $H_f = (5.86)(248/100) = 14.53$ ft (4.5 m).

4. Compute the total head on the pump.

The total head on the pump $H_t = H_s + H_f$. For the pump in Fig. 6-13a, $H_t = 110 + 14.53 = 124.53$ ft (38.0 m), say 125 ft (38.0 m). The total head on the pump in Fig. 6-13b and c would be the same. Some engineers term the total head on a pump the "total dynamic head" to distinguish between static head (no-flow vertical head) and operating head (rated flow through the pump).

The total head on the pumps in Fig. 6-13d, c , and f is computed in the same way as described above, except that the total static head is less because the pump has a static suction head—that is, the elevation of the liquid on the suction side reduces the total distance through which the pump must discharge liquid; thus the total static head is less. The static suction head is *subtracted* from the static discharge head to determine the total static head on the pump.

5. Compute the horsepower required to drive the pump.

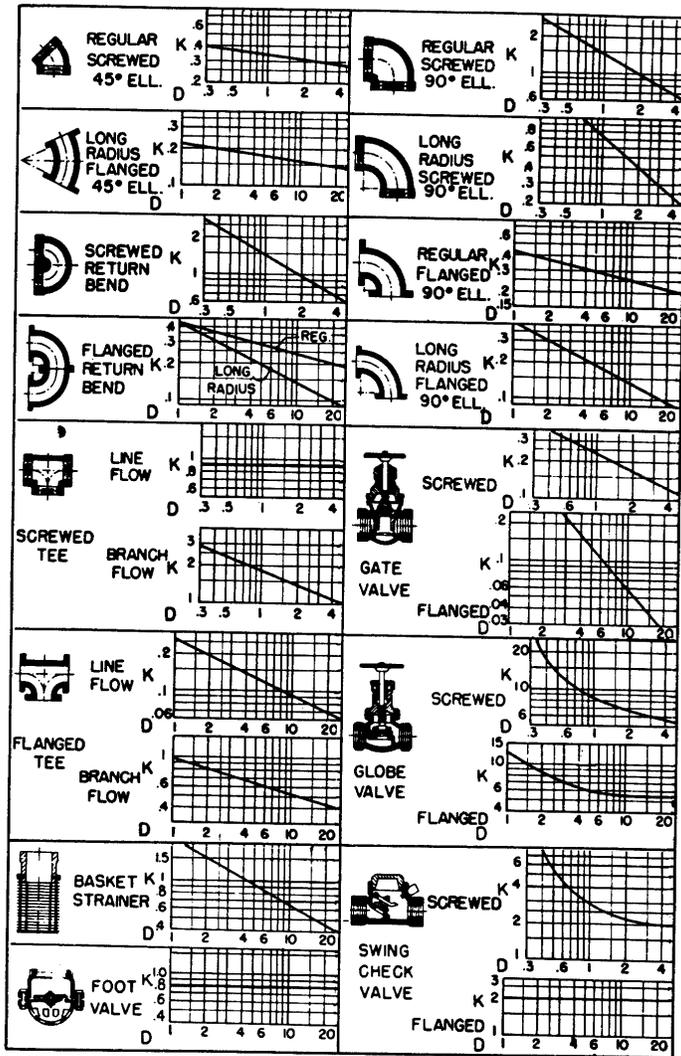
The brake horsepower input to a pump equals (gal/min)(H_t)(s)/3960 e , where $s =$ specific gravity of the liquid handled, and $e =$ hydraulic efficiency of the pump, expressed as a decimal. The usual hydraulic efficiency of a centrifugal pump is 60 to 80 percent;

Verification Case 58 Problem Statement

6-11

TABLE 6-22 Resistance of Fittings and Valves (length of straight pipe giving equivalent resistance)

Pipe size		Standard ell		Medium-radius ell		Long-radius ell		45° Ell		Tee		Gate valve, open		Globe valve, open		Swing check, open	
		ft	m	ft	m	ft	m	ft	m	ft	m	ft	m	ft	m	ft	m
6	152.4	16	4.9	14	4.3	11	3.4	7.7	2.3	33	10.1	3.5	1.1	160	48.8	40	12.2
8	203.2	21	6.4	18	5.5	14*	4.3	10	3.0	43	13.1	4.5	1.4	220	67.0	53	16.2
10	254.0	26	7.9	22	6.7	17	5.2	13	3.9	56	17.1	5.7	1.7	290	88.4	67	20.4
12	304.8	32	9.8	26	7.9	20	6.1	15	4.6	66	20.1	6.7	2.0	340	103.6	80	24.4



$$h = k \frac{v^2}{2g} \text{ feet of fluid.}$$

FIG. 6-14 Resistance coefficients of pipe fittings. (Hydraulic Institute.)

6-46 FLOW OF FLUIDS AND SOLIDS

TABLE 6-23 Pipe Friction Loss for Water (wrought-iron or steel schedule 40 pipe in good condition)

Diameter		Flow		Velocity		Velocity head		Friction loss/100 ft (30.5 m) pipe	
in	mm	gal/min	L/s	ft/s	m/s	ft water	m water	ft water	m water
6	152.4	1000	63.1	11.1	3.4	1.92	0.59	6.17	1.88
6	152.4	2000	126.2	22.2	6.8	7.67	2.3	23.8	7.25
6	152.4	4000	252.4	44.4	13.5	30.7	9.4	93.1	28.4
8	203.2	1000	63.1	6.41	1.9	0.639	0.195	1.56	0.475
8	203.2	2000	126.2	12.8	3.9	2.56	0.78	5.86	1.786
8	203.2	4000	252.4	25.7	7.8	10.2	3.1	22.6	6.888
10	254.0	1000	63.1	3.93	1.2	0.240	0.07	0.497	0.151
10	254.0	3000	189.3	11.8	3.6	2.16	0.658	4.00	1.219
10	254.0	5000	315.5	19.6	5.9	5.99	1.82	10.8	3.292

reciprocating pumps, 55 to 90 percent; rotary pumps, 50 to 90 percent. For each class of pump, the hydraulic efficiency decreases as the liquid viscosity increases.

Assume that the hydraulic efficiency of the pump in this system is 70 percent and the specific gravity of the liquid handled is 1.0. Then, input brake horsepower equals $(2000)(125)(1.0)/[3960(0.70)] = 90.2$ hp (67.4 kW).

The theoretical or hydraulic horsepower equals $(\text{gal/min})(H_t)(s)/3960 = (2000)(125)(1.0)/3960 = 64.1$ hp (47.8 kW).

Related Calculations: Use this procedure for any liquid—water, oil, chemical, sludge, etc.—whose specific gravity is known. When liquids other than water are being pumped, the specific gravity and viscosity of the liquid must be taken into consideration. The procedure given here can be used for any class of pump—centrifugal, rotary, or reciprocating.

Note that Fig. 6-14 can be used to determine the equivalent length of a variety of pipe fittings. To use Fig. 6-14, simply substitute the appropriate K value in the relation $h = Kv^2/2g$, where h = equivalent length of straight pipe; other symbols as before.

6-22 Pump Selection for Any Pumping System

Give a step-by-step procedure for choosing the class, type, capacity, drive, and materials for a pump that will be used in an industrial pumping system.

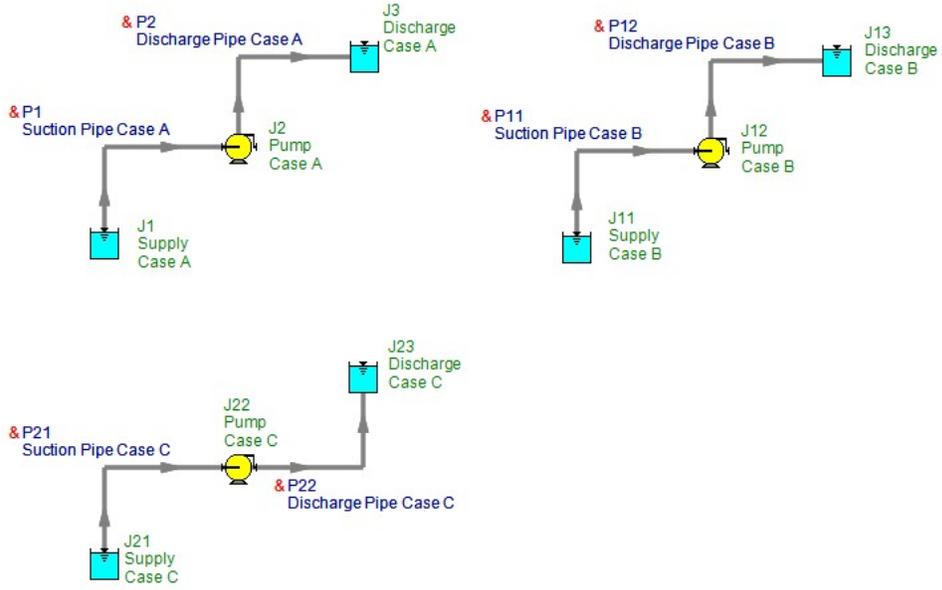
Calculation Procedure:

1. Sketch the proposed piping layout.

Use a single-line diagram (Fig. 6-15) of the piping system. Base the sketch on the actual job conditions. Show all the piping, fittings, valves, equipment, and other units in the system. Mark the actual and equivalent pipe length (see the previous example) on the sketch. Be certain to include all vertical lifts, sharp bends, sudden enlargements, storage tanks, and similar equipment in the proposed system.

View Verification Case 58 Model

[Verification Case 58](#)



Verification Case 59

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify59.fth

REFERENCE: Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 84-85

FLUID: Water

ASSUMPTIONS: Assume water at 70 deg. F.

RESULTS:

Pipe Flow Rate (ft ³ /sec)	1	2	3	4	5	6	7
Jeppson	0.841	0.666	1.334	0.573	0.761	0.175	0.398
AFT Fathom	0.8411	0.6664	1.3336	0.5726	0.761	0.1748	0.3979

Pipe Head Loss (feet)	1	2	3	4	5	6	7
Jeppson	3.15	17.63	15.46	1.5	11.42	0.674	3.24
AFT Fathom	*3.126	17.535	15.387	1.484	*11.374	0.664	*3.210

Node EGL (feet)	1	2	3	4
Jeppson	114.2	96.6	112	113.5
AFT Fathom	114.23	96.7	112.08	113.57

Pump Head (feet)	1	2	3
Jeppson	17.36	18.46	21.78
AFT Fathom	17.36	18.46	21.78

* AFT Fathom results combine two pipes, as discussed below

DISCUSSION:

Jeppson's method of applying pump data is to lump it into a pipe, whereas AFT Fathom's method is to place pumps at boundaries between pipes. Pumps are therefore a specific node (or junction) in AFT Fathom. To accommodate Jeppson's method, the pipe which contains the pump is split into two equivalent pipes in AFT Fathom. Where the split is made will have no impact on the results.

Because there are three pumps in the example, there are three additional pipes in the AFT Fathom model. AFT Fathom pipes 1 and 8 together represent Jeppson pipe 1. Similarly, AFT Fathom pipes 5 and 9 represent Jeppson pipe 5, and AFT Fathom pipes 7 and 10 represent Jeppson pipe 7.

Jeppson presents results in terms of HGL. However, Jeppson's method assumes EGL and HGL are essentially the same because of minimal velocity. Therefore, Jeppson results are presented in the results shown above as EGL.

The printed Jeppson results also give pressures at junctions/nodes 1-4. However, pressures can only be determined if elevation data is given for these locations, of which there is no information in the problem statement. Therefore no comparison is made to pressure.

Slight differences in property and calculation constants that were assumed, as well as potential differences from Jeppson's solution tolerances, which are not known, may also contribute to differences in the solution results. Examples are the specific value of water density and gravitational constant.

[List of All Verification Models](#)

Verification Case 59 Problem Statement

Verification Case 59

Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 84-85

Jeppson Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

84 PIPE NETWORK ANALYSES

work. The number of these pseudo loops must equal the difference between the number of unknown flow rates, i.e. N and $(J + L)$.

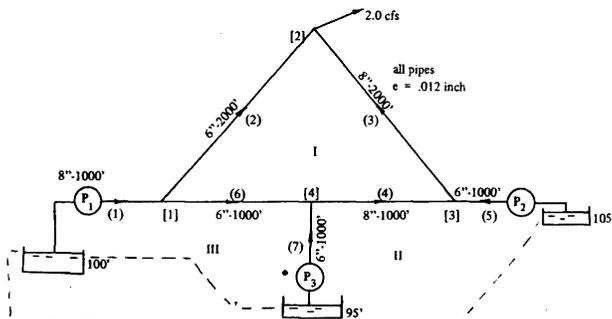
4. As many additional linear equations of the form $G \cdot Q = B/2A$ (Eq. 5-5) are written as pumps exist.
5. The nonlinear energy equations are linearized by defining coefficients K^1 of the Q 's which are obtained by $K^1 = KQ(m)^{n-1}$ and coefficients K_G^1 for the G unknowns are obtained by $K_G^1 = AG$.
6. The resulting system is solved iteratively, adjusting the coefficients as described earlier to reflect the average of the flow from the past two solutions until convergence occurs.

Should any of the details involved in these steps be vague, following them through for a simple example will be helpful. Consider the seven-pipe, one loop network supplied by three identical pumps shown below. Each pump supply head according to the equation

$$h_p = -10.328 Q_p^2 + 2.823 Q_p + 22.289$$

Since there are seven pipes in this network there will be seven unknown flow rates, plus three additional unknowns, i.e. the G 's of Eq. 5-5 for the three pumps which supply flow. Consequently a total of 10 simultaneous equations are needed. Four of these equations are the junction continuity equations; three are from Eq. 5-5 relating the three G 's to Q_1 , Q_3 , and Q_7 ; and consequently three energy equations are needed, one from the real loop and two from pseudo loops connecting pump reservoirs with no flow pipes. With the K 's in the exponential formula approximately computed by the Hazen-Williams equation, these equations are:

$$\text{Continuity Equations} \begin{cases} -Q_1 + Q_2 + Q_6 = 0 \\ -Q_2 - Q_3 = -2.0 \\ Q_3 - Q_4 - Q_5 = 0 \\ Q_4 - Q_6 - Q_7 = 0 \end{cases}$$



LINEAR THEORY METHOD 85

$$\text{Real Loop } \begin{cases} 55.7 Q_2^{1.85} - 13.7 Q_3^{1.85} - 6.86 Q_4^{1.85} \\ -27.8 Q_6^{1.85} = 0 \end{cases}$$

$$\text{Pseudo Loop } \begin{cases} 6.86 Q_1^{1.85} + 6.86 Q_4^{1.85} - 27.8 Q_5^{1.85} \\ + 27.8 Q_6^{1.85} + 10.33 G_1 - 10.33 G_2 = -5 \\ 6.86 Q_1^{1.85} + 27.8 Q_6^{1.85} - 27.8 Q_7^{1.85} \\ + 10.33 G_1 - 10.33 G_3 = 5 \end{cases}$$

$$\text{Transformation Eq. 5-5 } \begin{cases} -Q_1 + G_1 = -0.137 \\ -Q_5 + G_2 = -0.137 \\ -Q_7 + G_3 = -0.137 \end{cases}$$

In applying the linear theory method, the three nonlinear energy equations are linearized as described previously, and the resulting linear system solved. After three such iterative solutions of the linearized system the following solution results:

Pipe No.	1	2	3	4	5	6	7
Flow rate (cfs)	0.841	0.666	1.334	0.573	0.761	0.175	0.398
Head loss	3.15	17.63	15.46	1.50	11.42	0.674	3.24

Junction No.	1	2	3	4
Head (ft)	114.2	96.6	112.0	113.5
Pressure (psi)	49.5	41.9	48.5	49.2

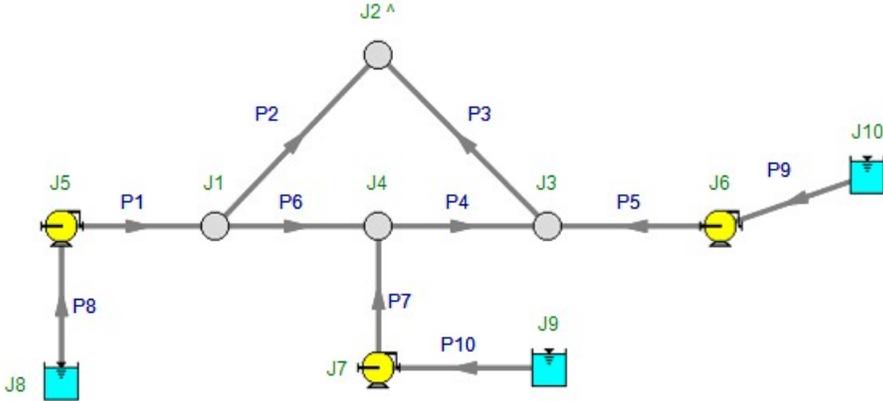
Pump	Head (ft)
1	17.36
2	18.46
3	21.78

**Including Pressure Reduction Valves
Into Linear Theory Method**

A pressure reducing valve (denoted PRV) is designed to maintain a constant pressure downstream from it regardless of how large the upstream pressure is. The exceptions to this occurrence are: (1) If the upstream pressure becomes less than the valve setting, and (2) if the downstream pressure exceeds the pressure setting of the valve so that if the PRV were not present the flow would be in the opposite direction to the downstream flow direction of the valve. If the first condition occurs, the valve has no effect on flow conditions. The PRV acts as a check valve, preventing reverse flow if the second condition occurs. By preventing

View Verification Case 59 Model

[Verification Case 59](#)



Verification Case 60

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify60.fth

REFERENCE: Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 86-87

FLUID: Water

ASSUMPTIONS: Assume water at 70 deg. F.

RESULTS:

Pipe Flow Rate (ft ³ /sec)	1	2	3	4	5	6	7
Jeppson	1.03	0.98	0.017	0.97	0.96	0.041	0.003
AFT Fathom	1.0268	0.984	0.016	0.973	0.960	0.039	0.003

Pipe Head Loss (feet)	1	2	3	4	5	6	7
Jeppson	23.35	21.43	0.009	4.21	40.92	0.05	23.68
AFT Fathom	*23.324	21.449	0.0065	4.194	40.829	*0.046	21.443

Node EGL (feet)	1	2	3	4
Jeppson	117.21	95.79	95.79	57.98
AFT Fathom	117.25	95.8	95.81	54.98

* AFT Fathom results combine two pipes, as discussed below

DISCUSSION:

Jeppson's method of applying pump and PRV data is to lump it into a pipe, whereas AFT Fathom's method is to place pumps and PRVs at boundaries between pipes. Pumps and PRVs are therefore a specific node (or junction) in AFT Fathom. To accommodate Jeppson's method, the pipe which contains the pump or PRV is split into two equivalent pipes in AFT Fathom. In the case of the pump, where the split is made will have no impact on the results. If the PRV control pressure is specified in terms of head, the elevation of the PRV becomes important. In such cases, Jeppson specifies the elevation and AFT Fathom incorporates this.

Because there is one pump and one PRV in the example, there are two additional pipes in the AFT Fathom model. AFT Fathom pipes 1 and 9 together represent Jeppson pipe 1. Similarly, AFT Fathom pipes 6 and 8 represent Jeppson pipe 6.

Jeppson presents results in terms of HGL. However, Jeppson's method assumes EGL and HGL are essentially the same because of minimal velocity. Therefore, Jeppson results are presented in the results shown above as EGL.

Results differ slightly between AFT Fathom and Jeppson for a few reasons. First, Jeppson represents pump curves differently than AFT Fathom. Jeppson typically uses an exponential formula (see page 82),

while AFT Fathom uses a polynomial based on a least squares curve fit. Second, the head loss formula used by Jeppson differs from AFT Fathom. Jeppson's formula is more common to the water industry, and assumes the head loss is proportional to flow rate to some power near but less than 2. AFT Fathom assumes it always proportional to flow rate to the power of 2. These differences affect the results to some degree.

Slight differences in property and calculation constants that were assumed, as well as potential differences from Jeppson's solution tolerances, which are not known, may also contribute to differences in the solution results. Examples are the specific value of water density and gravitational constant.

[List of All Verification Models](#)

Verification Case 60 Problem Statement

[Verification Case 60](#)

Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 86-87

[Jeppson Title Page](#)

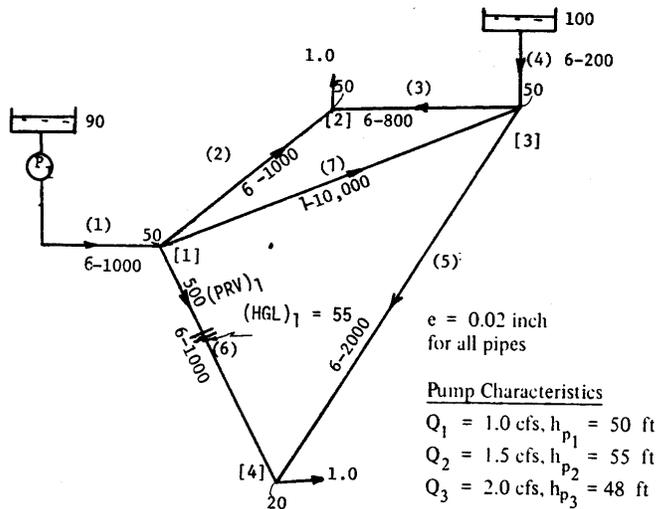
Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

86 PIPE NETWORK ANALYSES

reverse flow, the PRV allows the pressure immediately downstream from the valve to exceed its pressure setting. Thus, PRV's are used to reduce pressures in portions of a pipe distribution system if the pressures would otherwise be excessive, and they may also be used to control from which sources of supply the flow comes under various demand levels. In the latter applications the PRV acts as a check valve until the pressure is reduced to critical levels by large demands at which time additional sources of supply are drawn upon.

The analysis of a pipe network containing one or more PRV's must be capable of determining which of these conditions exist. Methods for accomplishing this which are consistent with the linear theory method are discussed in this section. Problems 11 through 15 in the next section provide additional examples, beyond those given in this section, of small networks containing PRV's.

Example No. 1 Methods for including PRV's in network analysis by the linear theory method will be illustrated by several examples. First consider the 7-pipe network shown below in which a PRV exists in pipe



6, 500 feet downstream from the beginning of this pipe. The system of Q-equations for this network consists of four junction continuity equations and three energy equations obtained from loops. The junction continuity equations are identical to those that would be written if the PRV were not present. In forming the loops for the energy equations, however, the pipe containing the PRV (pipe 6) is disconnected from its upstream junction

LINEAR THEORY METHOD 87

and the PRV is replaced by a reservoir. (This assumes the PRV reduces the pressure to a constant value. The PRV acting as a check valve is discussed later.) Of the three loop equations, one is obtained by summing the head losses around the real loop formed by pipes 2, 3, and 7. The second equation comes from the pseudo loop which connects the upper reservoir to the reservoir from which the pump obtains water (i.e., the source pump) by the pipes 4, 7, and 1. The third loop equation connects the artificial reservoir created by the PRV to another reservoir by a series of pipes such as 6, 5, and 4. Note that with pipe 6 disconnected from junction 1, only one real loop is available whereas two independent real loops existed before this. The real loop which is lost through the disconnection is compensated for by the additional pseudo loop from the artificial reservoir created by the PRV. Therefore, the number of equations which are available always equals the number of unknown flow rates Q . Using this scheme, the eight equations (eight are used instead of seven because one equation is added by the pump transformation as described in the previous section) needed for a solution by the linear theory method are:

$$\begin{array}{l}
 -Q_1 + Q_2 + Q_6 + Q_7 = 0.0 \\
 -Q_2 - Q_3 = -1.0 \\
 Q_3 - Q_4 + Q_5 - Q_7 = 0.0 \\
 -Q_5 - Q_6 = -1.0
 \end{array}
 \left. \vphantom{\begin{array}{l} \\ \\ \\ \end{array}} \right\} \begin{array}{l} \text{Junction} \\ \text{Continuity} \\ \text{Equations} \end{array}$$

$$\begin{array}{l}
 -K_2 Q_2^{n_2} + K_3 Q_3^{n_3} + K_7 Q_7^{n_7} = 0.0 \quad \text{Real Loop} \\
 K_4 Q_4^{n_4} - K_7 Q_7^{n_7} - K_1 Q_1^{n_1} + A_1 G_1^2 = 100 - 90 - h_{01} \quad \text{Pseudo Loop} \\
 G_1 - Q_1 = (B/2A)_1 \quad \text{Pump Transformation} \\
 K_6 Q_6^{n_6} - K_5 Q_5^{n_5} - K_4 Q_4^{n_4} = 55 - 100 \quad \text{Pseudo Loop from PRV}
 \end{array}$$

in which K_6 is determined only for the portion of pipe 6 downstream from the PRV.

Upon solving this system of equations by the linear theory method, using the procedure described previously, the following solution results:

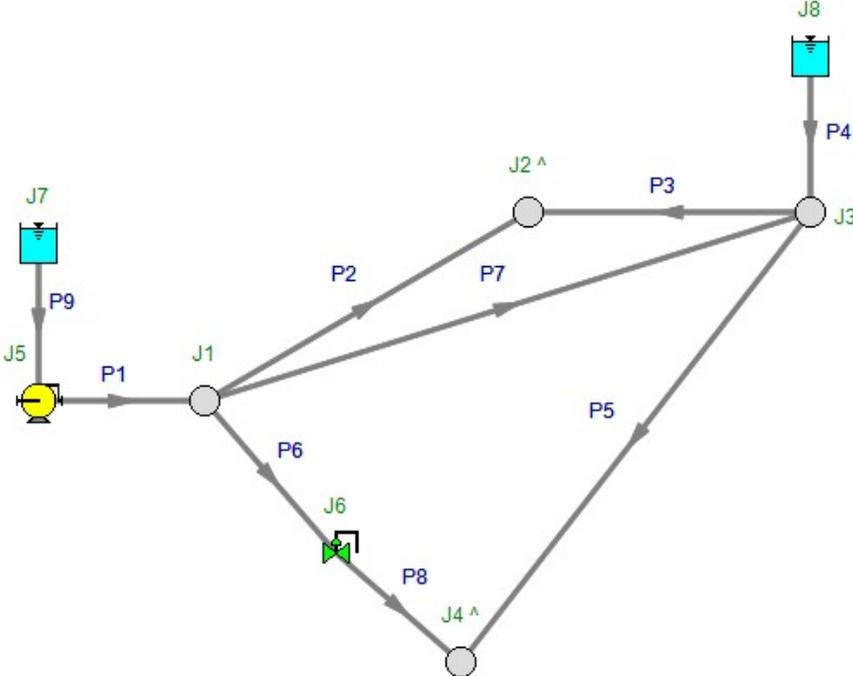
Pipe No.	1	2	3	4	5	6	7
Flow rate (cfs)	1.03	0.98	0.017	0.97	0.96	0.041	0.003
Head loss (ft)	23.35	21.43	0.009	4.21	40.92	0.05	23.68

The hydraulic grade line elevations at the junctions are:

Junction No.	1	2	3	4
Elevation of HGL (ft)	117.21	95.79	95.79	57.98

View Verification Case 60 Model

[Verification Case 60](#)



Verification Case 61

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify61.fth

REFERENCE: Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 135-137

FLUID: Water

ASSUMPTIONS: Assume water at 70 deg. F.

RESULTS:

Pipe Flow Rate (ft3/sec)	1	2	3	4	5	6	7	8
Jeppson	4.975	1.653	0.114	2.777	1.277	2.045	-1.039	-0.960
AFT Fathom	4.986	1.656	0.114	2.781	1.281	2.048	-1.042	-0.965

Pipe Flow Rate (ft3/sec)	9	10	11	12	13	14	15	16
Jeppson	1.415	0.889	0.609	0.109	0.413	1.587	2.391	0.025
AFT Fathom	1.413	0.891	0.608	0.108	0.411	1.589	2.395	0.014

Pipe Head Loss (feet)	1	2	3	4	5	6	7	8
Jeppson	11.32	10.60	0.03	8.41	2.22	13.25	4.48	1.83
AFT Fathom	*11.284	10.56	0.03	8.37	2.22	13.20	-4.48	-1.83

Pipe Head Loss (feet)	9	10	11	12	13	14	15	16
Jeppson	2.21	4.04	2.17	0.04	0.27	4.31	2.62	0.00
AFT Fathom	2.18	4.02	2.15	0.04	0.27	4.29	2.61	0.00

Node EGL (feet)	1	2	3	4	5	6	7	8
Jeppson	165.1	154.5	154.5	162.9	150.0	151.8	147.8	147.8
AFT Fathom	165.0	154.5	154.4	162.8	150.0	151.8	147.9	147.8

Node EGL (feet)	9
Jeppson	147.5
AFT Fathom	147.5

* AFT Fathom results combine two pipes, as discussed below

** Note that AFT Fathom represents head loss on pipes with reverse flow as a negative. Jeppson represents it as positive regardless of the direction.

DISCUSSION:

Jeppson's method of applying pump data is to lump it into a pipe, whereas AFT Fathom's method is to place pumps at boundaries between pipes. Pumps are therefore a specific node (or junction) in AFT Fathom. To accommodate Jeppson's method, the pipe which contains the pump is split into two equivalent pipes in AFT Fathom. Where the split is made will have no impact on the results.

Because there is one pump in the example, there is one additional pipe in the AFT Fathom model. AFT Fathom pipes 1 and 17 together represent Jeppson pipe 1.

Jeppson presents results in terms of HGL. However, Jeppson's method assumes EGL and HGL are essentially the same because of minimal velocity. Therefore, Jeppson results are presented in the results shown above as EGL.

Results differ slightly between AFT Fathom and Jeppson for a few reasons. First, Jeppson represents pump curves differently than AFT Fathom. Jeppson typically uses an exponential formula (see page 82), while AFT Fathom uses a polynomial based on a least squares curve fit. Second, the pipe head loss formula used by Jeppson differs from AFT Fathom. Jeppson's Hazen-Williams formula is given in his book and does not agree exactly with the accepted formula as used in AFT Fathom. These differences affect the results to some degree.

Slight differences in property and calculation constants that were assumed, as well as potential differences from Jeppson's solution tolerances, which are not known, may also contribute to differences in the solution results. Examples are the specific value of water density and gravitational constant.

Results for AFT Fathom also vary somewhat from previous versions of AFT Fathom (prior to version 7) because the equation used to convert the Hazen-Williams factor to the Darcy-Weisbach friction factor was modified to use the traditional formula, as given in the AFT Fathom help file.

[List of All Verification Models](#)

136 PIPE NETWORK ANALYSES

$$F_1 = K_2 (Q_{02} + \Delta Q_1 + \Delta Q_3)^{1.85} + K_3 (Q_{03} + \Delta Q_1 - \Delta Q_4)^{1.85} - K_4 (Q_{04} - \Delta Q_1)^{1.85} + K_5 (Q_{05} - \Delta Q_1)^{1.85} = 0$$

$$F_2 = -K_7 (Q_{07} - \Delta Q_2 - \Delta Q_3)^{1.85} + K_8 (Q_{08} + \Delta Q_2 - \Delta Q_5)^{1.85} - K_{15} (Q_{015} - \Delta Q_2)^{1.85} - K_3 (Q_{03} - \Delta Q_2 + \Delta Q_1)^{1.85} = 0$$

$$F_3 = K_2 (Q_{02} + \Delta Q_3 + \Delta Q_1)^{1.85} - K_7 (Q_{07} - \Delta Q_3 - \Delta Q_2)^{1.85} + K_8 (Q_{08} + \Delta Q_3 + \Delta Q_2)^{1.85} - K_6 (Q_{06} - \Delta Q_3)^{1.85} = 0$$

$$F_4 = K_{11} (Q_{011} + \Delta Q_4)^{1.85} + K_{12} (Q_{012} + \Delta Q_4)^{1.85} - K_9 (Q_{09} - \Delta Q_4 + \Delta Q_5)^{1.85} = 0$$

$$F_5 = K_9 (Q_{09} + \Delta Q_5 - \Delta Q_4)^{1.85} - K_{10} (Q_{010} - \Delta Q_5 + \Delta Q_6)^{1.85} - K_8 (Q_{08} - \Delta Q_5 + \Delta Q_2)^{1.85} = 0$$

$$F_6 = K_{10} (Q_{010} + \Delta Q_6 - \Delta Q_5)^{1.85} + K_{13} (Q_{013} + \Delta Q_6)^{1.85} - K_{14} (Q_{014} - \Delta Q_6)^{1.85} = 0$$

$$F_7 = K_{16} (Q_{016} + \Delta Q_7)^{1.85} + K_7 (Q_{07} + \Delta Q_7 - \Delta Q_2 - \Delta Q_3)^{1.85} - K_2 (Q_{02} - \Delta Q_7 + \Delta Q_2 + \Delta Q_3)^{1.85} - K_1 (Q_{01} - \Delta Q_7)^{1.85} + h_p - 150 = 0$$

In the last of these equations h_p is defined by the pump characteristic curve equation with $Q_{01} - \Delta Q_7$ replacing Q_p . After supplying an initial flow for each pipe, and solving by the Newton-Raphson method the following solution results:

Pipe No.	1	2	3	4	5	6	7
Q (cfs)	4.975	1.653	0.114	2.777	1.277	2.045	-1.039
h_f (ft)	11.32	10.60	0.03	8.41	2.22	13.25	4.48

NEWTON-RAPHSON METHOD 137

Pipe No.	8	9	10	11	12	13	14
Q (cfs)	-0.960	1.415	0.889	0.609	0.109	0.413	1.587
h_f (ft)	1.83	2.21	4.04	2.17	0.04	0.27	4.31

Pipe No.	15	16
Q (cfs)	2.391	0.025
h_f (ft)	2.62	0.00

Junction No.	1	2	3	4	5	6	7	8	9
Head (ft)	165.1	154.5	154.5	162.9	150.0	151.8	147.8	147.8	147.5

3. Solve the network shown below. The wall roughness for all pipes is $e = 0.012$ inch.

Solution:

Pipe No.	1	2	3	4	5	6	7
Q (cfs)	19.80	10.37	4.89	4.00	2.66	4.13	4.44
h_f (ft)	10.59	2.97	0.69	24.57	0.22	17.43	10.04

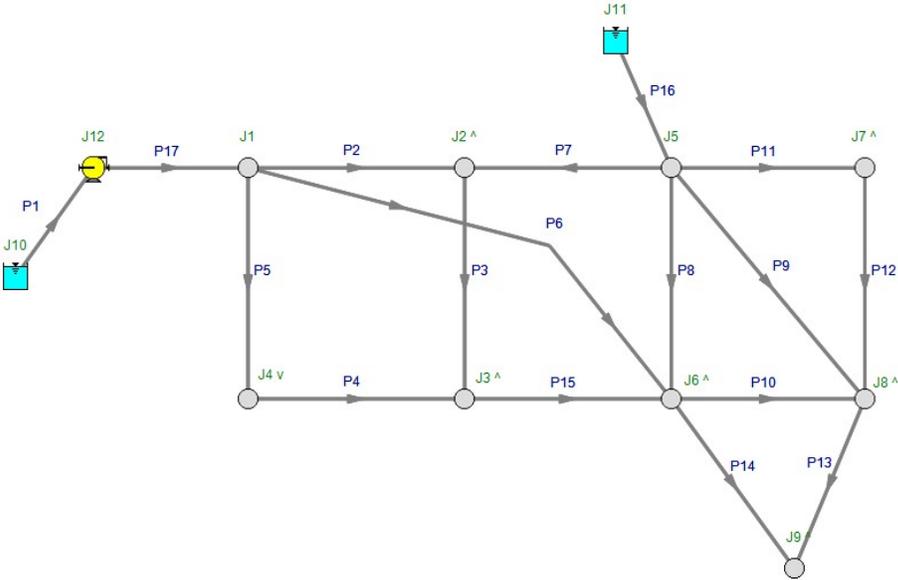
Pipe No.	8	9	10	11	12	13	14
Q (cfs)	4.60	13.67	2.40	4.07	6.07	1.64	1.11
h_f (ft)	16.17	11.30	3.01	16.90	2.29	2.86	1.01

Pipe No.	15	16	17	18	19	20	21
Q (cfs)	5.47	1.60	0.40	1.33	2.83	4.83	4.07
h_f (ft)	0.86	4.10	0.10	1.45	0.52	1.02	12.69

Pipe No.	22	23	24	25	26	27	28
Q (cfs)	2.59	1.41	3.07	3.11	3.07	2.57	6.98
h_f (ft)	10.46	1.07	7.29	15.00	7.28	5.16	61.39

View Verification Case 61 Model

[Verification Case 61](#)



Verification Case 62

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify62.fth

REFERENCE: Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 137-139

FLUID: Water

ASSUMPTIONS: Assume water at 70 deg. F.

RESULTS:

Pipe Flow Rate (ft3/sec)	1	2	3	4	5	6	7	8
Jeppson	19.8	10.37	4.89	4	2.66	4.13	4.44	4.6
AFT Fathom	19.798	10.375	4.891	4.003	2.656	4.131	4.439	4.601

Pipe Flow Rate (ft3/sec)	9	10	11	12	13	14	15	16
Jeppson	13.67	2.4	4.07	6.07	1.64	1.11	5.47	1.6
AFT Fathom	13.667	2.402	4.066	6.066	1.638	1.108	5.468	1.598

Pipe Flow Rate (ft3/sec)	17	18	19	20	21	22	23	24
Jeppson	0.4	1.33	2.83	4.83	4.07	2.59	1.41	3.07
AFT Fathom	0.402	1.336	2.836	4.836	4.068	2.591	1.409	3.066

Pipe Flow Rate (ft3/sec)	25	26	27	28	29	30	31	32
Jeppson	3.11	3.07	2.57	6.98	2.48	8.02	10.26	12.26
AFT Fathom	3.112	3.067	2.573	6.975	2.483	8.019	10.262	12.262

Pipe Flow Rate (ft3/sec)	33	34	35	36	37	38	39	40
Jeppson	2.94	1.06	17.21	0.63	8.06	11.69	4.5	12.03
AFT Fathom	2.944	1.056	17.206	0.635	8.064	11.699	4.494	12.033

Verification Case 62

Pipe Flow Rate (ft ³ /sec)	41	42	43	44	45	46	47	48
Jeppson	12.55	8.16	29.73	26.58	19.58	2.48	4.96	9.74
AFT Fathom	12.548	8.160	29.731	26.581	19.571	2.480	4.964	9.745

Pipe Head Loss (feet)	1	2	3	4	5	6	7	8
Jeppson	10.59	2.97	0.69	24.57	0.22	17.43	10.04	16.17
AFT Fathom	10.562	2.961	0.686	24.513	0.214	17.385	10.019	16.129

Pipe Head Loss (feet)	9	10	11	12	13	14	15	16
Jeppson	11.3	3.01	16.9	2.29	2.86	1.01	0.86	4.1
AFT Fathom	11.275	2.997	16.852	2.274	2.845	1.003	0.851	4.066

Pipe Head Loss (feet)	17	18	19	20	21	22	23	24
Jeppson	0.1	1.45	0.52	1.02	12.69	10.46	1.07	7.29
AFT Fathom	0.098	1.438	0.518	1.007	12.650	10.428	1.062	7.251

Pipe Head Loss (feet)	25	26	27	28	29	30	31	32
Jeppson	15	7.28	5.16	61.39	0.81	7.89	1.46	5.16
AFT Fathom	14.939	7.259	5.143	61.221	0.802	7.869	1.449	5.137

Pipe Head Loss (feet)	33	34	35	36	37	38	39	40
Jeppson	6.73	0.92	8.03	0.05	5.99	2.82	1.53	13.17
AFT Fathom	6.698	0.914	8.007	0.046	5.967	2.811	1.519	13.145

Pipe Head Loss (feet)	41	42	43	44	45	46	47	48
Jeppson	9.54	4.9	17.75	9.48	7.77	6.41	25.04	2.63
AFT Fathom	9.521	4.886	17.720	9.461	7.743	6.383	24.986	2.619

DISCUSSION:

The problem statement does not include any reservoirs or pressure junctions, and hence no EGL or pressure results can be obtained. Since AFT Fathom always displays EGL and pressure results, there must be a pressure. Thus junction 1 was chosen as a pressure junction and assigned a surface elevation of 200 feet. The particular value chosen does not affect the results.

Jeppson presents results in terms of HGL. However, Jeppson's method assumes EGL and HGL are essentially the same because of minimal velocity. Therefore, Jeppson results are presented in the results shown above as EGL.

Results differ slightly between AFT Fathom and Jeppson. The head loss formula used by Jeppson differs from AFT Fathom. Jeppson's formula is more common to the water industry, and assumes the head loss is proportional to flow rate to some power near but less than 2. AFT Fathom assumes it always proportional to flow rate to the power of 2. These differences affect the results to some degree.

Slight differences in property and calculation constants that were assumed, as well as potential differences from Jeppson's solution tolerances, which are not known, may also contribute to differences in the solution results. Examples are the specific value of water density and gravitational constant.

[List of All Verification Models](#)

Verification Case 62 Problem Statement

[Verification Case 62](#)

Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 137-139

[Jeppson Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

NEWTON-RAPHSON METHOD 137

Pipe No.	8	9	10	11	12	13	14
Q (cfs)	-0.960	1.415	0.889	0.609	0.109	0.413	1.587
h_f (ft)	1.83	2.21	4.04	2.17	0.04	0.27	4.31

Pipe No.	15	16
Q (cfs)	2.391	0.025
h_f (ft)	2.62	0.00

Junction No.	1	2	3	4	5	6	7	8	9
Head (ft)	165.1	154.5	154.5	162.9	150.0	151.8	147.8	147.8	147.5

3. Solve the network shown below. The wall roughness for all pipes is $e = 0.012$ inch.

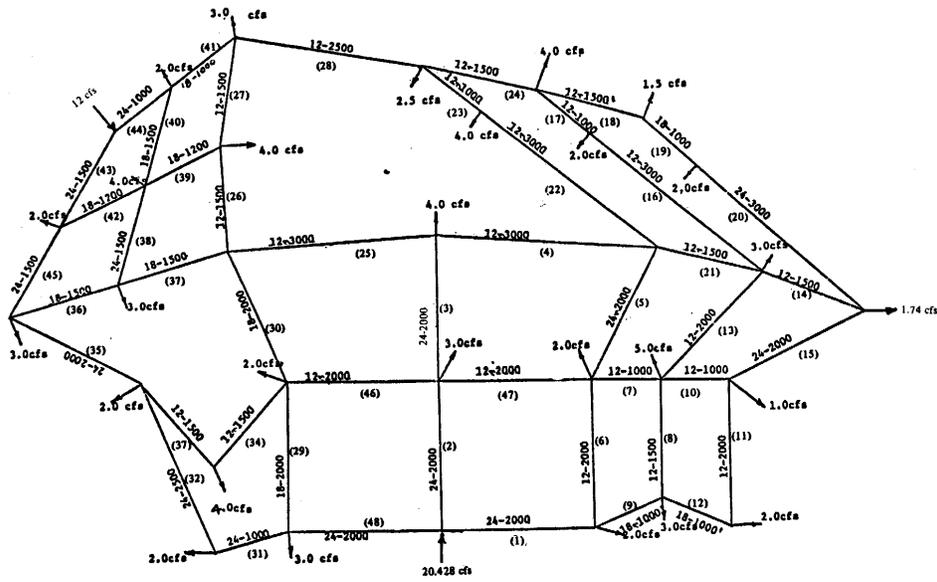
Solution:

Pipe No.	1	2	3	4	5	6	7
Q (cfs)	19.80	10.37	4.89	4.00	2.66	4.13	4.44
h_f (ft)	10.59	2.97	0.69	24.57	0.22	17.43	10.04

Pipe No.	8	9	10	11	12	13	14
Q (cfs)	4.60	13.67	2.40	4.07	6.07	1.64	1.11
h_f (ft)	16.17	11.30	3.01	16.90	2.29	2.86	1.01

Pipe No.	15	16	17	18	19	20	21
Q (cfs)	5.47	1.60	0.40	1.33	2.83	4.83	4.07
h_f (ft)	0.86	4.10	0.10	1.45	0.52	1.02	12.69

Pipe No.	22	23	24	25	26	27	28
Q (cfs)	2.59	1.41	3.07	3.11	3.07	2.57	6.98
h_f (ft)	10.46	1.07	7.29	15.00	7.28	5.16	61.39



NEWTON-RAPHSON METHOD 139

Pipe No.	29	30	31	32	33	34	35
Q (cfs)	2.48	8.02	10.26	12.26	2.94	1.06	17.21
h_f (ft)	0.81	7.89	1.46	5.16	6.73	0.92	8.03

Pipe No.	36	37	38	39	40	41	42
Q (cfs)	0.63	8.06	11.69	4.50	12.03	12.55	8.16
h_f (ft)	0.05	5.99	2.82	1.53	13.17	9.54	4.90

Pipe No.	43	44	45	46	47	48
Q (cfs)	29.73	26.58	19.58	2.48	4.96	9.74
h_f (ft)	17.75	9.48	7.77	6.41	25.04	2.63

4. Solve the 48-pipe network of example problem 3 using the Hazen-Williams formula assuming the coefficient $C_{HW} = 120$ for all pipes.

Solution:

Pipe No.	1	2	3	4	5	6	7
Q (cfs)	19.65	10.25	4.79	3.93	2.60	4.06	4.42
h_f (ft)	11.44	3.42	0.84	25.51	0.27	18.06	10.53

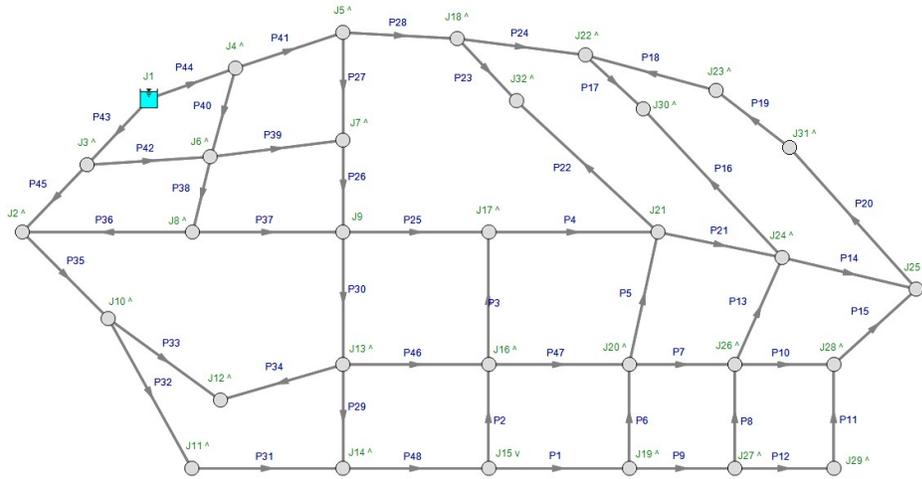
Pipe No.	8	9	10	11	12	13	14
Q (cfs)	4.58	13.59	2.39	4.01	6.01	1.61	1.09
h_f (ft)	16.87	11.72	3.37	17.64	2.59	3.23	1.78

Pipe No.	15	16	17	18	19	20	21
Q (cfs)	5.40	1.57	0.43	1.25	2.75	4.75	4.06
h_f (ft)	1.05	4.67	0.14	1.52	0.61	1.23	13.49

Pipe No.	22	23	24	25	26	27	28
Q (cfs)	2.48	1.52	3.18	3.14	3.04	2.47	7.20
h_f (ft)	10.88	1.46	8.60	16.83	7.93	5.39	65.07

View Verification Case 62 Model

[Verification Case 62](#)



Verification Case 63

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify63.fth

REFERENCE: Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 94

FLUID: Water

ASSUMPTIONS: Assume water at 70 deg. F.

RESULTS:

Pipe Flow Rate (ft ³ /sec)	1	2	3	4	5	6	7
Jeppson	0.533	0.662	1.338	0.699	0.639	-0.129	0.828
AFT Fathom	0.5352	0.6636	1.3417	0.7008	0.6409	-0.1284	0.8291

Pipe Head Loss (feet)	1	2	3	4	5	6	7
Jeppson	1.306	17.384	15.563	2.201	8.127	0.38	13.474
AFT Fathom	1.303	17.390	15.569	2.192	8.124	-0.372	*13.460

Node EGL (feet)	1	2	3	4
Jeppson	98.7	81.3	96.9	99.1
AFT Fathom	98.7	81.31	96.88	99.07

Node pressure (psig)	1	2	3	4
Jeppson	8.1	0.56	7.32	8.28
AFT Fathom	8.095	0.566	7.307	8.256

* AFT Fathom results combine two pipes, as discussed below

** Note that AFT Fathom represents head loss on pipes with reverse flow as a negative. Jeppson represents it as positive regardless of the direction.

DISCUSSION:

Jeppson's method of applying pump data is to lump it into a pipe, whereas AFT Fathom's method is to place pumps at boundaries between pipes. Pumps are therefore a specific node (or junction) in AFT Fathom. To accommodate Jeppson's method, the pipe which contains the pump is split into two equivalent pipes in AFT Fathom. Where the split is made will have no impact on the results.

Because there is one pump in the example, there is one additional pipe in the AFT Fathom model. AFT Fathom pipes 7 and 8 together represent Jeppson pipe 7.

Jeppson presents results in terms of HGL. However, Jeppson's method assumes EGL and HGL are essentially the same because of minimal velocity. Therefore, Jeppson results are presented in the results shown above as EGL.

Slight differences in property and calculation constants that were assumed, as well as potential differences from Jeppson's solution tolerances, which are not known, may also contribute to differences in the solution results. Examples are the specific value of water density and gravitational constant.

[List of All Verification Models](#)

Verification Case 63 Problem Statement

Verification Case 63

Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 94

Jeppson Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

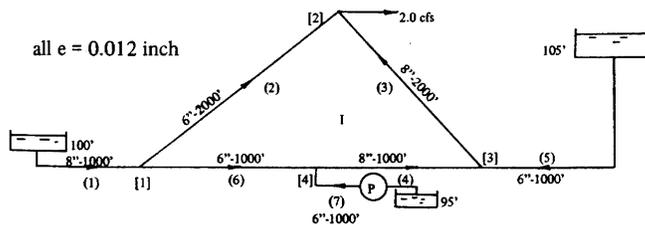
LINEAR THEORY METHOD 93

Pipe No.	6	7	8	9	10
Flow rate (cfs)	1.432	0.326	0.174	1.758	2.242
Head loss (ft)	38.15	3.86	0.85	12.83	20.70

PRV No.	1	2	3
Upstream elevation of HGL (ft)	448.21	446.23	485.67
Downstream elevation of HGL (ft)	150.00	150.00	150.00

Example Problems Which Include Pumps and Reservoirs

- Water supply comes from one pump and two reservoirs as shown in the sketch. How many pseudo loops need to be established? Write the system of equations whose solution provides the flow rate in each pipe using symbols K and n in the energy equations. The pump characteristic curve is given by: $h_p = -10.33 Q_p^2 + 2.823 Q_p + 22.29$.



Solution:

Two pseudo loops are required. A possibility is one pseudo loop connecting the reservoirs supplying pipes 1 and 5 through pipes 1, 6, 4, and 5; and the other connects the pump reservoir and the reservoir supplying pipe 1 through pipes 1, 6, and 7.

$$\text{Continuity Equations} \begin{cases} -Q_1 + Q_2 + Q_6 = 0 \\ -Q_2 - Q_3 = -2.0 \\ Q_3 - Q_4 - Q_5 = 0 \\ Q_4 - Q_6 - Q_7 = 0 \end{cases}$$

$$\text{Real Loop} \quad \{ K_2 Q_2^{n_2} - K_3 Q_3^{n_3} - K_4 Q_4^{n_4} - K_6 Q_6^{n_6} = 0$$

94 PIPE NETWORK ANALYSES

$$\text{Pseudo Loop} \quad \begin{cases} K_1 Q_1^{n_1} + K_6 Q_6^{n_6} + K_4 Q_4^{n_4} - K_5 Q_5^{n_5} = -5 \\ K_1 Q_1^{n_1} + K_6 Q_6^{n_6} - K_7 Q_7^{n_7} + 10.33 G_1^2 \\ \quad \quad \quad = -5 + 22.36 \end{cases}$$

$$\text{Transformation Equation} \quad \begin{cases} -Q_7 + G_1 = -0.137 \end{cases}$$

2. Solve the network of problem 1 giving the flow rates in each pipe, the head loss in each pipe, and the head and pressures at each junction, if the elevation of all junctions is at 80 ft.

Solution:

Pipe No.	1	2	3	4	5	6	7
Flow rate (cfs)	0.533	0.662	1.338	0.699	0.639	-0.129	0.828
Head loss (ft)	1.306	17.384	15.563	2.201	8.127	0.380	13.474

Junction No.	1	2	3	4
Elev. HGL	98.7	81.3	96.9	99.1
Pressure (psi)	8.10	0.56	7.32	8.28

3. Solve problem 1 if the pump is removed from pipeline 7 and the system is supplied by the three reservoirs.

Solution:

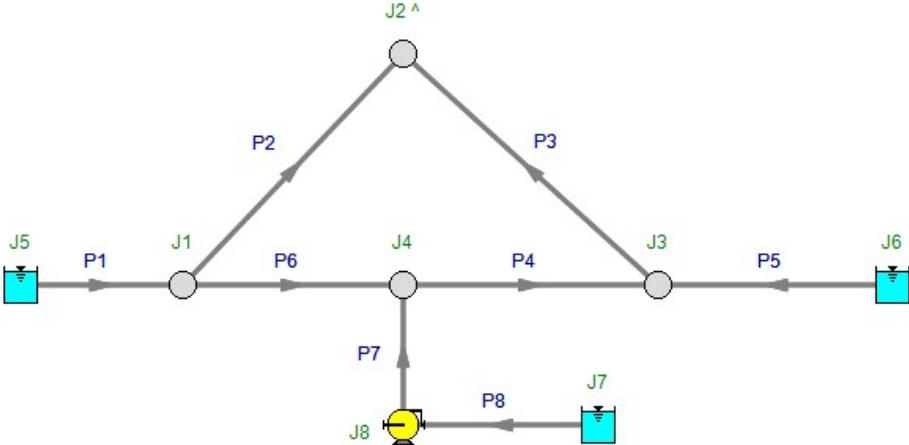
To solve this system there will only be seven unknowns instead of the eight as in problems 1 and 2, since the pump does not introduce an additional unknown G . The four continuity equations and the first two energy equations are identical to those given in the solution to problem 1. The final energy equation is,

$$K_1 Q_1^{n_1} + K_6 Q_6^{n_6} - K_7 Q_7^{n_7} = -5$$

The solution by the linear theory method produces the following after three iterative solutions.

View Verification Case 63 Model

[Verification Case 63](#)



Verification Case 64

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify64.fth

REFERENCE: Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 109-110

FLUID: Water

ASSUMPTIONS: Assume water at 70 deg. F.

RESULTS:

Pipe Flow Rate (ft ³ /sec)	1	2	3	4	5	6	7	8
Jeppson	2.56	-0.32	2.44	0.73	0.88	1.12	1.83	3.17
AFT Fathom	2.53	-0.38	2.47	0.72	0.92	1.08	1.81	3.19

Pipe Head Loss (feet)	1	2	3	4	5	6	7	8
Jeppson	130.61	1.89	119.67	8.75	18.42	24.12	6.33	17.88
AFT Fathom	*128.35	** -2.66	122.19	8.50	*19.98	22.64	6.25	17.75

PRV EGL (feet)	Up	Down
Jeppson	53.8	50.0
AFT Fathom	49.9	49.9

* AFT Fathom results combine two pipes, as discussed below

** Note that AFT Fathom represents head loss on pipes with reverse flow as a negative. Jeppson represents it as positive regardless of the direction.

DISCUSSION:

Jeppson's method of applying pump and PRV data is to lump it into a pipe, whereas AFT Fathom's method is to place pumps and PRVs at boundaries between pipes. Pumps and PRVs are therefore a specific node (or junction) in AFT Fathom. To accommodate Jeppson's method, the pipe which contains the pump or PRV is split into two equivalent pipes in AFT Fathom. In the case of the pump, where the split is made will have no impact on the results. If the PRV control pressure is specified in terms of head, the elevation of the PRV becomes important. In such cases, Jeppson specifies the elevation and AFT Fathom incorporates this.

Because there is one pump and one PRV in the example, there are two additional pipes in the AFT Fathom model. AFT Fathom pipes 1 and 10 together represent Jeppson pipe 1. Similarly, AFT Fathom pipes 5 and 9 represent Jeppson pipe 5.

Jeppson presents results in terms of HGL. However, Jeppson's method assumes EGL and HGL are essentially the same because of minimal velocity. Therefore, Jeppson's results are presented in the results shown above as EGL.

Results differ slightly between AFT Fathom and Jeppson for a few reasons. First, Jeppson represents pump curves differently than AFT Fathom. Jeppson typically uses an exponential formula (see page 82), while AFT Fathom uses a polynomial based on a least squares curve fit. Second, the pipe head loss formula used by Jeppson differs from AFT Fathom. Jeppson's Hazen-Williams formula is given in his book and does not agree exactly with the accepted formula as used in AFT Fathom. These differences affect the results to some degree.

Slight differences in property and calculation constants that were assumed, as well as potential differences from Jeppson's solution tolerances, which are not known, may also contribute to differences in the solution results. Examples are the specific value of water density and gravitational constant.

Results for AFT Fathom also vary somewhat from previous versions of AFT Fathom (prior to version 7) because the equation used to convert the Hazen-Williams factor to the Darcy-Weisbach friction factor was modified to use the traditional formula, as given in the AFT Fathom help file.

Because of the slight differences in calculations between AFT Fathom and Jeppson, there is slightly less pressure head available across the PRV than the 50 feet specified in the problem statement. Thus, there is a warning message generated in AFT Fathom that the control valve is unable to control, and has failed open. Results are displayed above for the failed open case. AFT Fathom shows warnings in the Warnings section at the top of the Output window. In addition, the Valve Summary at the top of the Output window shows the PRV status.

[List of All Verification Models](#)

Verification Case 64 Problem Statement

[Verification Case 64](#)

Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 109-110

[Jeppson Title Page](#)

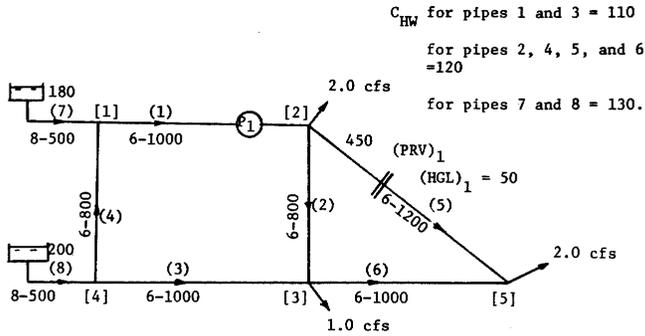
Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

LINEAR THEORY METHOD 109

Junction	HGL (m)	Head (m)	Pressure (N/cm ²)
1	308.6	8.6	8.43
2	255.0	5.0	4.92
3	262.2	7.2	7.03
4	289.6	29.6	29.0
5	281.1	1.12	1.09
6	281.3	1.34	1.31
7	349.5	9.5	9.31
8	286.4	16.4	16.1
9	339.4	3.4	9.19
10	377.5	17.5	17.1
11	260.4	0.45	0.44
12	425.9	5.93	5.81
13	275.9	5.88	5.76
14	259.6	15.6	15.3
15	284.9	24.9	24.4
16	273.8	13.8	13.5
17	274.6	14.6	14.3
18	302.2	22.2	21.7
19	277.7	7.7	7.56
20	315.5	15.5	15.2
21	270.3	20.3	19.9
22	268.4	8.4	8.25
23	210.4	10.4	10.2
24	174.2	14.2	14.0
25	171.3	11.3	11.0
26	88.3	18.3	17.9
27	81.3	11.3	11.0
28	45.1	5.1	5.0
29	45.2	25.2	24.7
30	284.8	24.8	24.4
31	230.1	30.1	29.5
32	-37.5	12.5	12.2
33	-51.8	8.2	8.0

11. Solve the 8-pipe network shown in the accompanying sketch which is supplied by two reservoirs with water surfaces at 180 and 200 ft, respectively. The PRV is located 450 ft downstream from the upstream junction of pipe 5 and has a pressure setting which maintains the elevation of the HGL at 50 ft downstream from the valve.

110 PIPE NETWORK ANALYSES



Pump Characteristics

$Q_{p_1} = 1.0 \text{ cfs}, h_{p_1} = 40.0'$, $Q_{p_2} = 1.5 \text{ cfs}, h_{p_2} = 35.0'$, $Q_{p_3} = 2.0 \text{ cfs}, h_{p_3} = 26.0'$

Solution:

Pipe No.	1	2	3	4
Flow rate (cfs)	2.56	-0.32	2.44	0.73
Head loss (ft)	130.61	1.89	119.67	8.75

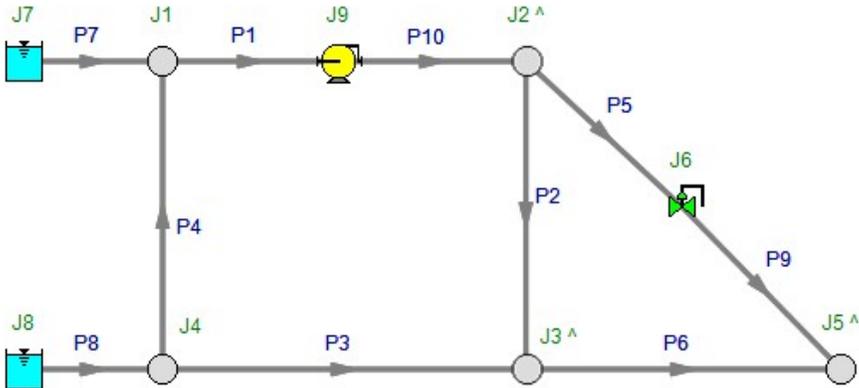
Pipe No.	5	6	7	8
Flow rate (cfs)	0.88	1.12	1.83	3.17
Head loss (ft)	18.42	24.12	6.33	17.88

HGL upstream from PRV = 53.80 ft
HGL downstream from PRV = 50.00 ft

12. Obtain the flow rates in each pipe, and the pressure at each junction of the network in the accompanying sketch. Water is flowing (in SI units the kinematic viscosity $\nu = 0.0000113 \text{ N}^2/\text{s} = 0.0133 \text{ Stokes}$, and the specific weight $\gamma = 9800 \text{ N/m}^3$).

View Verification Case 64 Model

[Verification Case 64](#)



Verification Case 65

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify65.fth

REFERENCE: Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 110-111

FLUID: Water

ASSUMPTIONS: Assume water at 15 deg. C.

RESULTS:

Pipe Flow Rate (m3/sec)	1	2	3	4	5	6	7	8
Jeppson	0.6402	0.2857	0.1857	-0.0857	0.2344	-0.2402	0	0
AFT Fathom	0.651	0.282	0.182	-0.082	0.249	-0.251	0.000	0.000

Pipe Flow Rate (m3/sec)	9	10
Jeppson	0.12	-0.1902
AFT Fathom	0.120	-0.201

Pipe Head Loss (m)	1	2	3	4	5	6	7	8
Jeppson	6	16.21	12.32	1.79	30.32	11.77	0	0
AFT Fathom	6.132	15.664	11.798	-1.662**	29.124	-12.640**	0.000*	0.000*

Pipe Head Loss (m)	9	10
Jeppson	56.47	1.91
AFT Fathom	55.766	-2.104**

Node EGL (meters)	1	2	3	4	5	6
Jeppson	294	277.79	265.48	263.68	251.91	237.54
AFT Fathom	293.9	278.2	266.4	264.7	252.1	238.1

PRV EGL (meters)	1 Up	1 Down	2 Up	2 Down
Jeppson	251.91	237.54	265.48	237.54
AFT Fathom	252.1	238.1	266.4	238.1

* AFT Fathom results combine two pipes, as discussed below

** Note that AFT Fathom represents head loss on pipes with reverse flow as a negative. Jeppson represents it as positive regardless of the direction.

DISCUSSION:

Jeppson's method of applying PRV data is to lump it into a pipe, whereas AFT Fathom's method is to place PRVs at boundaries between pipes. PRVs are therefore a specific node (or junction) in AFT Fathom. To accommodate Jeppson's method, the pipe which contains the PRV is split into two equivalent pipes in AFT Fathom. If the PRV control pressure is specified in terms of head, the elevation of the PRV becomes important. In such cases, Jeppson specifies the elevation and AFT Fathom incorporates this.

Because there are two PRVs in the example, there are two additional pipes in the AFT Fathom model. AFT Fathom pipes 7 and 11 together represent Jeppson pipe 7. Similarly, AFT Fathom pipes 8 and 12 represent Jeppson pipe 8.

Jeppson presents results in terms of HGL. However, Jeppson's method assumes EGL and HGL are essentially the same because of minimal velocity. Therefore, Jeppson results are presented in the results shown above as EGL.

In both Jeppson's solution and AFT Fathom's, the two PRVs cannot control to their set pressure head, and fail closed. Results are displayed above for the failed closed case. AFT Fathom shows warnings in the Warnings section at the top of the Output window. In addition, the Valve Summary at the top of the Output window shows the PRV status.

Results differ slightly between AFT Fathom and Jeppson. The head loss formula used by Jeppson differs from AFT Fathom. Jeppson's formula is more common to the water industry, and assumes the head loss is proportional to flow rate to some power near but less than 2. AFT Fathom assumes it always proportional to flow rate to the power of 2. These differences affect the results to some degree.

Slight differences in property and calculation constants that were assumed, as well as potential differences from Jeppson's solution tolerances, which are not known, may also contribute to differences in the solution results. Examples are the specific value of water density and gravitational constant.

[List of All Verification Models](#)

Verification Case 65 Problem Statement

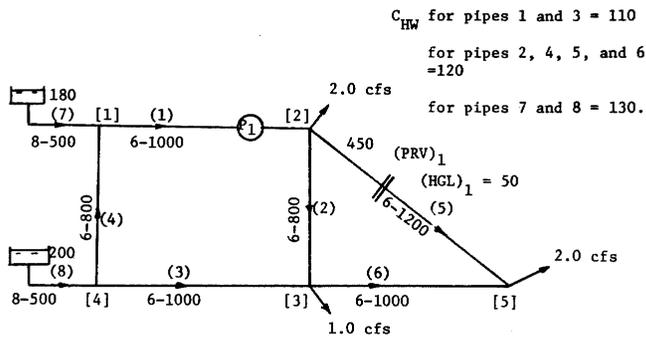
Verification Case 65

Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 110-111

Jeppson Title Page

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

110 PIPE NETWORK ANALYSES



Pump Characteristics

$$Q_{p_1} = 1.0 \text{ cfs}, h_{p_1} = 40. \text{'}; Q_{p_2} = 1.5 \text{ cfs}, h_{p_2} = 35. \text{'}; Q_{p_3} = 2.0 \text{ cfs}, h_{p_3} = 26. \text{'}$$

Solution:

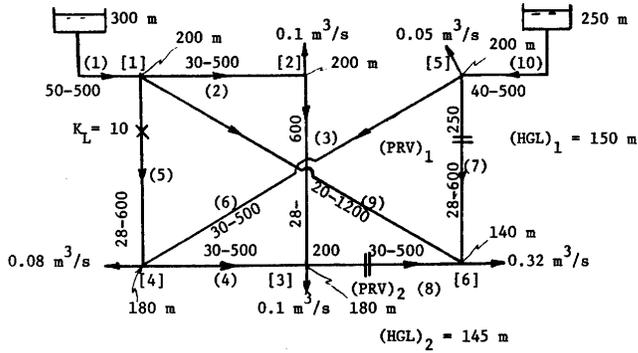
Pipe No.	1	2	3	4
Flow rate (cfs)	2.56	-0.32	2.44	0.73
Head loss (ft)	130.61	1.89	119.67	8.75

Pipe No.	5	6	7	8
Flow rate (cfs)	0.88	1.12	1.83	3.17
Head loss (ft)	18.42	24.12	6.33	17.88

HGL upstream from PRV = 53.80 ft
 HGL downstream from PRV = 50.00 ft

12. Obtain the flow rates in each pipe, and the pressure at each junction of the network in the accompanying sketch. Water is flowing (in SI units the kinematic viscosity $\nu = 0.00000113 \text{ N}^2/\text{s} = 0.0133 \text{ Stokes}$, and the specific weight $\gamma = 9800 \text{ N/m}^3$).

LINEAR THEORY METHOD 111



Solution:

Pipe No.	1	2	3	4	5
Flow rate m ³ /s	0.6402	0.2857	0.1857	-0.0857	0.2344
Head loss (m)	6.00	16.21	12.32	1.79	30.32

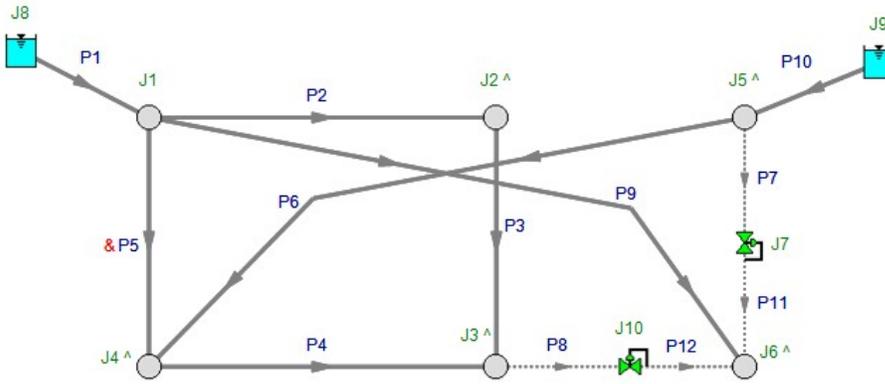
Pipe No.	6	7	8	9	10
Flow rate m ³ /s	-0.2402	0.0	0.0	0.12	-0.1902
Head loss (m)	11.77	0.0	0.0	56.47	1.91

Junction No.	1	2	3	4	5	6
Elev. of HGL	294.0	277.79	265.48	263.68	251.91	237.54
Pressure (kPa)						

PRV No.	1	2
Upstream, HGL (m)	251.91	265.48
Downstream, HGL (m)	237.54	237.54

View Verification Case 65 Model

[Verification Case 65](#)



Verification Case 66

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify66.fth

REFERENCE: Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 88-89

FLUID: Water

ASSUMPTIONS: Assume water at 70 deg. F.

RESULTS:

Pipe Flow Rate (ft ³ /sec)	1	2	3	4	5	6	7	8
Jeppson	0.5	0.37	2.16	3.16	0	1	0.25	0.25
AFT Fathom	0.500	0.374	2.168	3.168*	0.000*	1.000	0.246	0.254

Pipe Flow Rate (ft ³ /sec)	9	10	11	12	13	14
Jeppson	0.05	1	0.79	0.71	0.13	1.84
AFT Fathom	0.046	1.000	0.794	0.706	0.126	1.832

Pipe Head Loss (feet)	1	2	3	4	5	6	7	8
Jeppson	2.44	0.63	19.27	6.4	0	6.4	1.01	1.06
AFT Fathom	2.409	0.636	19.217	6.375*	0.000*	6.329	0.989	1.045

Pipe Head Loss (feet)	9	10	11	12	13	14
Jeppson	0.06	2.56	2.73	2.22	0.09	2.21
AFT Fathom	0.056	2.532	2.699	2.148	0.085	2.179

PRV EGL (feet)	Up	Down
Jeppson	417.6	393.04
AFT Fathom	417.6	393.1

* AFT Fathom results combine two pipes, as discussed below

** The problem statement has a typo on pipe length. It says 1000 ft., and it should only be 500 ft.

DISCUSSION:

Jeppson's method of applying pump and PRV data is to lump it into a pipe, whereas AFT Fathom's method is to place pumps and PRVs at boundaries between pipes. Pumps and PRVs are therefore a

specific node (or junction) in AFT Fathom. To accommodate Jeppson's method, the pipe which contains the pump or PRV is split into two equivalent pipes in AFT Fathom. In the case of the pump, where the split is made will have no impact on the results. If the PRV control pressure is specified in terms of head, the elevation of the PRV becomes important. In such cases, Jeppson specifies the elevation and AFT Fathom incorporates this.

Because there is one pump and one PRV in the example, there are two additional pipes in the AFT Fathom model. AFT Fathom pipes 4 and 16 together represent Jeppson pipe 4. Similarly, AFT Fathom pipes 5 and 15 represent Jeppson pipe 5.

In both Jeppson's solution and AFT Fathom's, the PRV cannot control to its set pressure head, and it fails closed. Results are displayed above for the failed closed case. AFT Fathom shows warnings in the Warnings section at the top of the Output window. In addition, the Valve Summary at the top of the Output window shows the PRV status.

Results differ slightly between AFT Fathom and Jeppson for a few reasons. First, Jeppson represents pump curves differently than AFT Fathom. Jeppson typically uses an exponential formula (see page 82), while AFT Fathom uses a polynomial based on a least squares curve fit. Second, the head loss formula used by Jeppson differs from AFT Fathom. Jeppson's formula is more common to the water industry, and assumes the head loss is proportional to flow rate to some power near but less than 2. AFT Fathom assumes it always proportional to flow rate to the power of 2. These differences affect the results to some degree.

Slight differences in property and calculation constants that were assumed, as well as potential differences from Jeppson's solution tolerances, which are not known, may also contribute to differences in the solution results. Examples are the specific value of water density and gravitational constant.

[List of All Verification Models](#)

Verification Case 66 Problem Statement

[Verification Case 66](#)

Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 88-89

[Jeppson Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

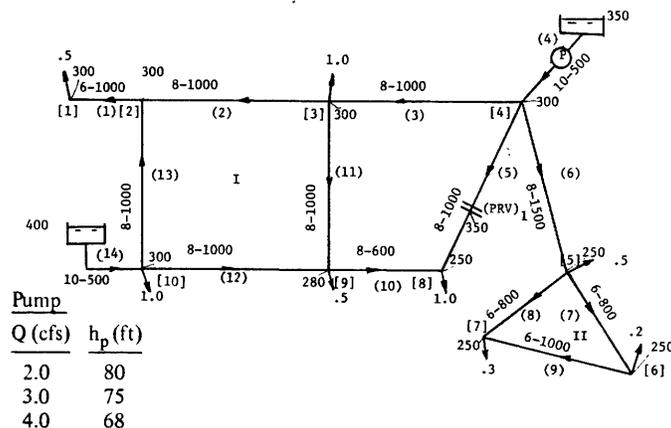
88 PIPE NETWORK ANALYSES

The pressure upstream from the PRV equals 117.19 ft. and downstream equals 55 ft. Consequently, the assumption used in writing the final loop equation above is correct. Had the solution given a negative flow rate in pipe 6, this assumption would be incorrect, since the PRV would then have acted as a check valve and allowed the elevation of the HGL downstream from the PRV to rise above 55 ft. Should this have occurred, then the flow rate in pipe 6 would no longer be unknown, but equal to zero. Under these conditions caused by the PRV shutting flow off, the final equation above would be replaced by:

$$\dots(HGL)_1 - K_5 Q_5^{n_5} - K_4 Q_4^{n_4} = -100$$

From the above example, it should be evident how networks containing PRV's might be analyzed by the linear theory method. The assumption is initially made that the pressure (or the elevation of the HGL) immediately downstream from the PRV is constant and equal to the valve setting. The junction continuity equations are written as if no PRV's existed. To write the loop equations, pipes containing PRV's are disconnected from their upstream junctions and the PRV's are replaced by reservoirs. The real and pseudo loops are established for this modified network. After each iteration a check of the flow Q in each pipe containing a PRV is made. Should any of these flows be negative, then the pseudo loop equation which includes terms for that pipe is altered, replacing that Q as an unknown with the HGL immediately downstream from that PRV.

Example No. 2 As an example of a network in which a PRV acts as a check valve, consider the 14-pipe network in the sketch below. In this network, pipe 5 contains a PRV with a pressure setting which results in



LINEAR THEORY METHOD 89

the elevation of the HGL downstream from the valve equaling 350 ft. For this network 10 linear junction continuity equations are available, and two energy equations around the real loops formed by pipes 2, 11, 12, and 13, and 8, 9, and 7, respectively. One additional energy equation is available from a pseudo loop connecting the reservoir to the source pump through a series of pipes such as 14, 13, 2, 3, and 4. The tenth and final equation is obtained by connecting the artificial reservoir created by the PRV to the real reservoir or source pump by a series of pipes such as 5, 10, 12, and 14. This final equation is,

$$K_5 Q_5^{n_5} - K_{10} Q_{10}^{n_{10}} - K_{12} Q_{12}^{n_{12}} - K_{14} Q_{14}^{n_{14}} = (HGL)_1 - 400$$

Initially, $(HGL)_1$ might be assumed known and equal to 350 ft. If this is done, however, the linear theory method immediately produces a negative flow rate, Q_5 , from the first iteration. Consequently $Q_5 = 0$ and the above equation should be rewritten with $(HGL)_1$ unknown as,

$$-(HGL)_1 - K_{10} Q_{10}^{n_{10}} - K_{12} Q_{12}^{n_{12}} - K_{14} Q_{14}^{n_{14}} = -400$$

The solution obtained using this equation as the fourteenth equation of the system of equations is

Pipe No.	1	2	3	4	5	6	7
Flow rate (cfs)	0.50	0.37	2.16	3.16	0.0	1.00	0.25
Head loss (ft)	2.44	0.63	19.27	6.40	0.0	6.40	1.01

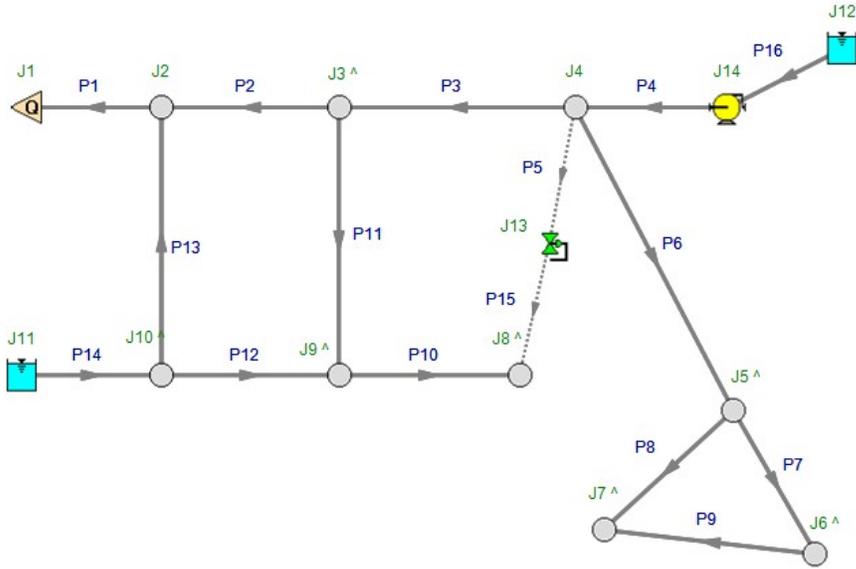
Pipe No.	8	9	10	11	12	13	14
Flow rate (cfs)	0.25	0.05	1.00	0.79	0.71	0.13	1.84
Head loss (ft)	1.06	0.06	2.56	2.73	2.22	0.09	2.21

The elevations of the HGL upstream and downstream from the PRV are 417.60 ft and $(HGL)_1 = 393.04$ ft, respectively.

While it is possible to obtain a solution to a network such as this 14-pipe network in a single analysis, it might be analyzed piecemeal more efficiently. Since all the demands at junctions 5, 6, and 7 must come through pipe 6, its flow rate can be determined readily as the sum of these demands or $Q_6 = 1.0$ cfs. Now the subnetwork consisting of pipes 7, 8, and 9 can be disconnected from the remainder of the network and each subnetwork analyzed separately. When large distribution systems are involved, considerable reduction in storage requirements as well as computational efficiency can result by disconnecting the network into subnetworks where possible.

View Verification Case 66 Model

[Verification Case 66](#)



Verification Case 67

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify67.fth

REFERENCE: Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 95-98

FLUID: Water

ASSUMPTIONS: Assume water at 60 deg. F.

RESULTS:

Pipe Flow Rate (ft3/sec)	1	2	3	4	5	6	7	8
Jeppson	2.94	-1.76	-0.54	1.74	0.88	-2.55	-3.35	2.17
AFT Fathom	2.94	-1.76	-0.54	1.74	0.88	-2.56	-3.36	2.18

Pipe Flow Rate (ft3/sec)	9	10	11	12	13	14	15	16
Jeppson	3.07	-0.44	-0.58	0.64	0.73	1.32	1.18	0.80
AFT Fathom	3.07	-0.44	-0.58	0.64	0.73	1.32	-1.19	0.80

Pipe Flow Rate (ft3/sec)	17	18	19	20	21	22	23	24
Jeppson	-2.29	-0.17	0.09	3.27	2.45	-0.04	1.15	-0.41
AFT Fathom	-2.29	-0.17	0.09	3.28	2.46	-0.04	1.15	-0.42

Pipe Flow Rate (ft3/sec)	25	26	27	28
Jeppson	6.84	6.01	3.35	-2.39
AFT Fathom	6.85	6.02	3.34	-2.40

Pipe Head Loss (feet)	1	2	3	4	5	6	7	8
Jeppson	6.41	11.85	0.62	9.10	10.76	30.20	45.50	51.10
AFT Fathom	6.38	-11.80	-0.61	9.02	10.62	-30.11	-45.32	50.87

Pipe Head Loss (feet)	9	10	11	12	13	14	15	16
Jeppson	43.90	7.92	9.90	8.20	9.38	59.00	29.60	23.40
AFT Fathom	43.73	-7.87	-9.86	8.15	9.29	58.74	-29.47	23.28

Verification Case 67

Pipe Head Loss (feet)	17	18	19	20	21	22	23	24
Jeppson	7.86	1.37	0.28	37.60	30.70	0.01	6.90	6.91
AFT Fathom	-7.82	-1.32	0.27	37.46	30.56	-0.01	6.89	-6.90

Pipe Head Loss (feet)	25	26	27	28
Jeppson	30.6	0.91	0.31	8.43
AFT Fathom	*30.517	*0.90	*0.304	-8.437

Node EGL (feet)	1	2	3	4	5	6	7	8
Jeppson	1365	1359	1347	1348	1357	1346	1316	1270
AFT Fathom	1365.2	1358.8	1347.0	1347.7	1356.7	1346.1	1315.9	1270.6

Node EGL (feet)	9	10	11	12	13	14	15	16
Jeppson	1321	1329	1339	1347	1392	1354	1361	1354
AFT Fathom	1321.5	1329.4	1339.2	1347.4	1392.0	1354.5	1361.4	1354.5

* AFT Fathom results combine two pipes, as discussed below

** Note that AFT Fathom represents head loss on pipes with reverse flow as a negative. Jeppson represents it as positive regardless of the direction.

DISCUSSION:

Jeppson's method of applying pump data is to lump it into a pipe, whereas AFT Fathom's method is to place pumps at boundaries between pipes. Pumps are therefore a specific node (or junction) in AFT Fathom. To accommodate Jeppson's method, the pipe which contains the pump is split into two equivalent pipes in AFT Fathom. Where the split is made will have no impact on the results.

Because there are three pumps in the example, there are three additional pipes in the AFT Fathom model. AFT Fathom pipes 25 and 29 together represent Jeppson pipe 25. Similarly, AFT Fathom pipes 26 and 31 represent Jeppson pipe 26, and AFT Fathom pipes 27 and 30 represent Jeppson pipe 27.

Jeppson presents results in terms of HGL. However, Jeppson's method assumes EGL and HGL are essentially the same because of minimal velocity. Therefore, Jeppson results are presented in the results shown above as EGL.

Results differ slightly between AFT Fathom and Jeppson for a few reasons. First, Jeppson represents pump curves differently than AFT Fathom. Jeppson typically uses an exponential formula (see page 82), while AFT Fathom uses a polynomial based on a least squares curve fit. Second, the head loss formula used by Jeppson differs from AFT Fathom. Jeppson's formula is more common to the water industry, and

assumes the head loss is proportional to flow rate to some power near but less than 2. AFT Fathom assumes it always proportional to flow rate to the power of 2. These differences affect the results to some degree.

Slight differences in property and calculation constants that were assumed, as well as potential differences from Jeppson's solution tolerances, which are not known, may also contribute to differences in the solution results. Examples are the specific value of water density and gravitational constant.

Results for AFT Fathom vary somewhat from previous versions of AFT Fathom (prior to version 7) because the equation used to convert the Hazen-Williams factor to the Darcy-Weisbach friction factor was modified to use the traditional formula, as given in the AFT Fathom help file.

[List of All Verification Models](#)

Verification Case 67 Problem Statement

[Verification Case 67](#)

Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 95-98

[Jeppson Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

LINEAR THEORY METHOD 95

Pipe No.	1	2	3	4	5	6	7
Q (cfs)	0.979	0.680	1.320	0.527	0.793	0.300	0.227
h_f (ft)	4.23	18.32	15.16	1.28	12.39	1.88	1.11

4. Solve the single real loop pipe network supplied by the three pumps described just before these example problems by the Hazen-Williams equation assuming all pipes have a coefficient $C_{HW} = 120$.

Solution:

The equations are the same as previously given with the exception that the n 's = 1.852, and the K 's are constant and given by: $K_1 = 4.85$, $K_2 = 39.3$, $K_3 = 9.69$, $K_4 = 4.85$, $K_5 = 19.7$, $K_6 = 19.7$, $K_7 = 19.7$. The solution is

Pipe No.	1	2	3	4	5	6	7
Q (cfs)	0.784	0.681	1.313	0.564	0.751	0.257	0.368
h_f (ft)	3.09	19.3	16.1	1.68	11.6	1.59	3.09

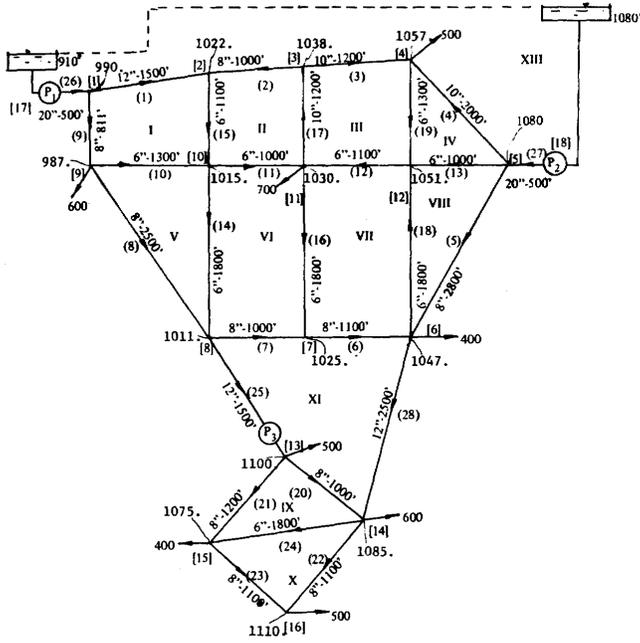
5. Solve problem 3 using the Hazen-Williams formula with all $C_{HW} = 120$.

Solution:

Pipe No.	1	2	3	4	5	6	7
Q (cfs)	0.965	0.684	1.316	0.522	0.795	0.281	0.241
h_f (ft)	4.53	19.45	16.13	1.45	12.86	1.87	1.41

6. Obtain a solution to the 28 pipe network below by the linear theory method through use of a program such as the one whose input is described in Appendix C.

96 PIPE NETWORK ANALYSES



Pipe No.	C_{HW}	Pipe No.	C_{HW}
1	130	15	120
2	130	16	120
3	120	17	130
4	120	18	120
5	120	19	120
6	120	20	130
7	120	21	120
8	120	22	120
9	100	23	120
10	100	24	120
11	100	25	130
12	130	26	110
13	130	27	110
14	120	28	120

LINEAR THEORY METHOD 97

Pump Characteristics

$$\begin{aligned} \text{Pump No. 1 } h_p &= -0.417 Q_p^2 + 5.14 Q_p + 440.3 \\ \text{Pump No. 2 } h_p &= -0.378 Q_p^2 - 5.09 Q_p + 298.2 \\ \text{Pump No. 3 } h_p &= -2.51 Q_p^2 + 16.7 Q_p + 155.3 \end{aligned}$$

Data for program:

```
O PROBLEM 6, CHAPTER 5
$$SPECIF NPIPE=28,NRES=0,NPUMP=3,NBPUMP=1,NFLOW=1,COEFRO=120.,
NPLENG=1 $END
27
1080.
26 27 25
5.568 455.97 7.0156 455.81 7.795 455. 910.
2.004 280.5 2.3385 284.25 3.341 277. 1080.
4.009 182. 4.454 180. 4.9 177. 0.
1 12. 1.5 130. 1 2 1022.
2 8. 1. 130. 3 2
3 10. 1.2 3 4 500. 1057
4 10. 2. 5 4
5 8. 2.8 5 6 400. 1047.
6 8. 1.1 7 6
7 8. 1. 8 7
8 8. 2.5 9 8 1011.
9 8. .811 100. 1 9 600. 987.
10 6. 1.3 100. 9 10 1015.
11 6. 1. 100. 10 11 700. 1030.
12 6. 1.1 130. 12 11
13 6. 1. 130. 5 12 1051.
14 6. 1.8 10 8
15 6. 1.1 2 10
16 6. 1.8 11 7 1025.
17 10. 1.2 130. 11 3 1038.
18 6. 1.8 6 12
19 6. 1.3 4 12
20 8. 1. 130. 13 14 600. 1085.
21 8. 1.2 13 15 400. 1075.
22 8. 1.1 14 16 500. 1110.
23 8. 1.1 15 16
24 6. 1.8 14 15
25 12. 1.5 130. 8 13 500. 1100
26 20. .5 110. 1 990.
27 20. .5 110. 5 1080.
28 12. 2.5 6 14
END
```

Solution:

One pseudo loop must be defined which connects the reservoirs from which pumps 1 and 2 obtain their supplies, giving 16 junction continuity equations and 12 energy equations. In addition three linear transformation equations are added for the three pumps in the network. The data required by the program in Appendix C follows. The solution is provided below.

98 PIPE NETWORK ANALYSES

Pipe No.	1	2	3	4	5	6	7
Q (cfs)	2.94	-1.76	-0.54	1.74	0.88	-2.55	-3.35
h_f (ft)	6.41	11.85	0.62	9.10	10.76	30.2	45.5

Pipe No.	8	9	10	11	12	13	14
Q (cfs)	2.17	3.07	-0.44	-0.58	0.64	0.73	1.32
h_f (ft)	51.1	43.9	7.92	9.90	8.20	9.38	59.0

Pipe No.	15	16	17	18	19	20	21
Q (cfs)	1.18	0.80	-2.29	-0.17	0.09	3.27	2.45
h_f (ft)	29.6	23.4	7.86	1.37	0.28	37.6	30.7

Pipe No.	22	23	24	25	26	27	28
Q (cfs)	-0.04	1.15	-0.41	6.84	6.01	3.35	-2.39
h_f (ft)	0.01	6.90	6.91	30.6	0.91	0.31	8.43

Junction No.	1	2	3	4	5	6	7	8
Elev. HGL (ft)	1365	1359	1347	1348	1357	1346	1316	1270

Junction No.	9	10	11	12	13	14	15	16
Elev. HGL (ft)	1321	1329	1339	1347	1392	1354	1361	1354

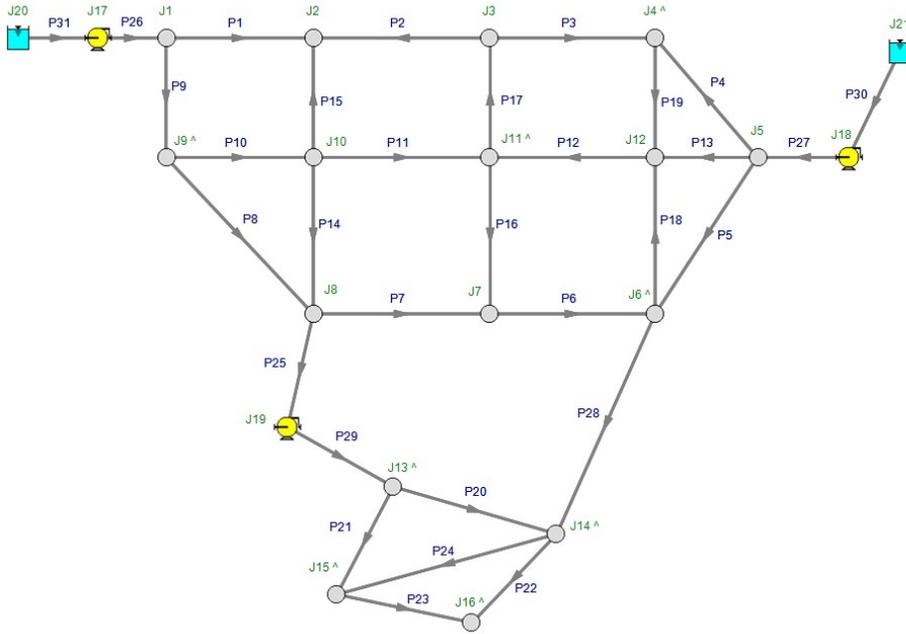
7. What are the flow rates and head losses, as well as the elevation of the HGL in the network of problem 6 if pipeline 28 is taken out of operation. If the elevation at junctions are as given below in one of the tables of solutions what is the pressure at each junction. The solution should be obtained by the computer program such as those whose input is described in Appendix C.

Solution:

With pipeline 28 removed a single line connects two separate networks, each of which might be solved as separate problems. Obviously, if the analysis were being done by hand two separate analyses would be used and subsequently tied together. With the computer, however, it is more convenient to consider only one network containing 27 pipes, 16 junctions, and 10 real loops. The additional energy equation comes from

View Verification Case 67 Model

[Verification Case 67](#)



Verification Case 68

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify68.fth

REFERENCE: Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 105-109

FLUID: Water

ASSUMPTIONS: Assume water at 22 deg. C.

RESULTS:

Pipe Flow Rate (m3/sec)	1	2	3	4	5	6	7	8
Jeppson	0.0869	0.0208	0.0132	0.0153	0.0092	0.0274	0.0254	0.0148
AFT Fathom	0.087	0.021	0.013	0.015	0.009	0.027	0.026	0.015

Pipe Flow Rate (m3/sec)	9	10	11	12	13	14	15	16
Jeppson	-0.0027	0.0299	0.0212	0.0081	0.0089	0.0164	0.0273	0.0107
AFT Fathom	-0.002	0.031	0.022	0.008	0.007	0.019	-0.026	0.011

Pipe Flow Rate (m3/sec)	17	18	19	20	21	22	23	24
Jeppson	0.0118	0.0078	0.0243	0.0791	0.0351	-0.0035	0.0106	0.0166
AFT Fathom	0.012	0.007	0.025	0.079	0.035	-0.003	-0.010	0.016

Pipe Flow Rate (m3/sec)	25	26	27	28	29	30	31	32
Jeppson	0.0119	0.0277	0.0805	-0.0043	0.0061	-0.0065	0.0148	0.0071
AFT Fathom	0.012	0.028	0.081	-0.004	0.006	-0.006	0.015	0.007

Pipe Flow Rate (m3/sec)	33	34	35	36	37	38	39	40
Jeppson	0.0046	0.0274	0.0094	0.0219	0.1306	-0.0214	-0.0299	-0.0238
AFT Fathom	0.005	0.027	0.009	0.022	0.130	-0.021	-0.030	-0.024

Verification Case 68

Pipe Flow Rate (m3/sec)	41	42	43	44	45	46	47	48
Jeppson	0.0239	0.0287	0.045	0.0139	-0.0002	0.0056	0.0051	0.0167
AFT Fathom	0.024	0.029	0.045	0.014	0.000	0.006	0.005	0.017

Pipe Flow Rate (m3/sec)	49	50	51	52	53	54	55	56
Jeppson	0.0137	-0.0078	0.0222	0.0547	0.021	0.0165	0.0135	-0.002
AFT Fathom	0.014	-0.008	0.022	0.055	0.021	0.016	0.013	-0.002

Pipe Flow Rate (m3/sec)	57	58	59	60	61	62	63
Jeppson	0.0116	0.0109	0.0031	0.0072	0.0078	0.0003	0.1276
AFT Fathom	0.012	0.011	0.003	0.007	0.008	0.000	0.127

Pipe Head Loss (meters)	1	2	3	4	5	6	7	8
Jeppson	201.1	53.6	19.1	27.4	7.16	27.5	27.3	8.3
AFT Fathom	*201.95	53.59	19.23	27.33	7.02	27.42	27.31	8.08

Pipe Head Loss (meters)	9	10	11	12	13	14	15	16
Jeppson	0.22	68.4	63.1	5.11	10.1	38.1	91	78.9
AFT Fathom	-0.11	71.74	67.04	4.60	6.94	22.29	-82.39	83.02

Pipe Head Loss (meters)	17	18	19	20	21	22	23	24
Jeppson	165.5	86.5	48.4	166.9	150	16.3	117.7	45.8
AFT Fathom	156.70	73.67	51.38	*164.52	146.50	-14.69	-109.81	44.37

Pipe Head Loss (meters)	25	26	27	28	29	30	31	32
Jeppson	29.7	25.3	287.6	1.37	12.52	3.74	11.16	6.62
AFT Fathom	29.68	26.17	*289.26	-1.25	12.24	-3.35	10.99	6.82

Pipe Head Loss (meters)	33	34	35	36	37	38	39	40
Jeppson	0.8	27.6	3.5	31.8	297.8	12.7	24.4	15.7
AFT Fathom	0.82	27.31	3.47	31.60	299.65	-12.41	-24.29	-15.45

Verification Case 68

Pipe Head Loss (meters)	41	42	43	44	45	46	47	48
Jeppson	37.8	45.2	54.6	11.1	0.11	55	54.9	267.5
AFT Fathom	37.41	44.72	*54.40	10.98	-0.01	55.85	55.84	266.16

Pipe Head Loss (meters)	49	50	51	52	53	54	55	56
Jeppson	322.4	14.3	336.7	240.7	94.2	58.1	39.1	2.97
AFT Fathom	322.00	-14.12	336.10	*242.34	93.82	57.84	38.90	-2.92

Pipe Head Loss (meters)	57	58	59	60	61	62	63
Jeppson	93	82.9	7.07	36.2	43.2	0.12	284.5
AFT Fathom	92.57	82.65	7.01	35.99	42.89	0.11	286.52

Node EGL (meters)	1	2	3	4	5	6	7	8
Jeppson	308.6	255	262.2	289.6	281.1	281.3	349.5	286.4
AFT Fathom	307.17	253.58	260.61	287.94	279.75	279.86	351.50	284.46

Node EGL (meters)	9	10	11	12	13	14	15	16
Jeppson	339.4	377.5	260.4	425.9	275.9	259.6	284.9	273.8
AFT Fathom	344.56	366.85	261.54	418.23	271.73	257.04	283.21	272.22

Node EGL (meters)	17	18	19	20	21	22	23	24
Jeppson	274.6	302.2	277.7	315.5	270.3	268.4	210.4	174.2
AFT Fathom	273.04	300.35	276.06	313.48	268.75	266.57	208.73	172.75

Node EGL (meters)	25	26	27	28	29	30	31	32
Jeppson	171.3	88.3	81.3	45.1	45.2	284.8	230.1	-37.5
AFT Fathom	169.83	87.18	80.18	44.18	44.29	283.20	227.36	-38.80

Node EGL (meters)	33
Jeppson	-51.8
AFT Fathom	-52.92

* AFT Fathom results combine two pipes, as discussed below

** Note that AFT Fathom represents head loss on pipes with reverse flow as a negative. Jeppson represents it as positive regardless of the direction.

DISCUSSION:

Jeppson's method of applying pump data is to lump it into a pipe, whereas AFT Fathom's method is to place pumps at boundaries between pipes. Pumps are therefore a specific node (or junction) in AFT Fathom. To accommodate Jeppson's method, the pipe which contains the pump is split into two equivalent pipes in AFT Fathom. Where the split is made will have no impact on the results.

Because there are five pumps in the example, there are five additional pipes in the AFT Fathom model. AFT Fathom pipes 1 and 64 together represent Jeppson pipe 1. Similarly, AFT Fathom pipes 20 and 65 represent Jeppson pipe 20, AFT Fathom pipes 27 and 66 represent Jeppson pipe 27, AFT Fathom pipes 43 and 68 represent Jeppson pipe 43, and AFT Fathom pipes 52 and 67 represent Jeppson pipe 52.

Jeppson presents results in terms of HGL. However, Jeppson's method assumes EGL and HGL are essentially the same because of minimal velocity. Therefore, Jeppson results are presented in the results shown above as EGL.

Results differ slightly between AFT Fathom and Jeppson for a few reasons. First, Jeppson represents pump curves differently than AFT Fathom. Jeppson typically uses an exponential formula (see page 82), while AFT Fathom uses a polynomial based on a least squares curve fit. Second, the head loss formula used by Jeppson differs from AFT Fathom. Jeppson's formula is more common to the water industry, and assumes the head loss is proportional to flow rate to some power near but less than 2. AFT Fathom assumes it always proportional to flow rate to the power of 2. These differences affect the results to some degree. Third, this system is highly networked which may cause some individual pipes, especially those with lower flow rates, to differ quite a bit from AFT Fathom. Looking at the system as a whole, the agreement is quite good between Jeppson and AFT Fathom.

Slight differences in property and calculation constants that were assumed, as well as potential differences from Jeppson's solution tolerances, which are not known, may also contribute to differences in the solution results. Examples are the specific value of water density and gravitational constant.

[List of All Verification Models](#)

Verification Case 68 Problem Statement

[Verification Case 68](#)

Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 105-109

[Jeppson Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

LINEAR THEORY METHOD 105

Junction No.	1	2	3	4	5	6
HGL (ft)	390.2	376.0	366.7	363.9	366.6	363.1
Pressure (psi)	104.1	97.9	89.6	92.7	87.3	88.0

Junction No.	7	8	9	10	11	12
HGL (ft)	374.2	372.3	376.1	369.1	368.9	363.2
Pressure (psi)	84.2	79.0	76.3	81.9	77.5	75.0

Junction No.	13	14	15	16	17	18
HGL (ft)	345.0	341.8	362.0	333.8	340.9	343.3
Pressure (psi)	71.5	78.8	91.9	79.6	87.1	83.8

Junction No.	19	20	21	22	23	24
HGL (ft)	348.1	364.9	351.5	347.3	342.4	340.8
Pressure (psi)	77.2	80.1	78.65	81.2	83.4	82.7

Junction No.	25	26	27	28	29
HGL (ft)	319.5	316.1	342.3	346.1	347.3
Pressure (psi)	77.8	80.7	92.0	93.6	96.3

Using the linear theory program described in Appendix C, the solution was obtained in six iterations with a sum of changes in flow rate equal to 0.019. The solution requires 23.3 seconds of execution time on a UNIVAC 1108.

- Determine the flow rates and head losses of each of the 63 pipes of the network shown below. The diameter of each pipe is given in centimeters and the length of each pipe is given in meters. Thus pipe number 1 is 16 cm in diameter and has a length of 1500 m. All 63 pipes of the network have a wall roughness of 0.026 cm. The network is supplied by 2 reservoirs without pumps and 4 pumps obtaining their water from the reservoirs as shown in the sketch. A booster pump exists in pipe 43 also. The water surface elevations are as given on the sketch in meters and the pump characteristic curves are defined by the data for three points as given below.

106 PIPE NETWORK ANALYSES

Pump No.	Point 1		Point 2		Point 3	
	Q m ³ /s	h _p (m)	Q m ³ /s	h _p (m)	Q m ³ /s	h _p (m)
1	0.03	84.	0.045	70.	0.06	52.
2	0.06	45.	0.105	39.	0.15	31.
3	0.06	45.	0.105	39.	0.15	31.
4	0.06	45.	0.105	39.	0.15	31.
5	0.03	9.0	0.045	7.5	0.06	5.5

The elevation of the junctions are as given below. What is the elevation of the HGL at each junction, the head in meters, and the pressure?

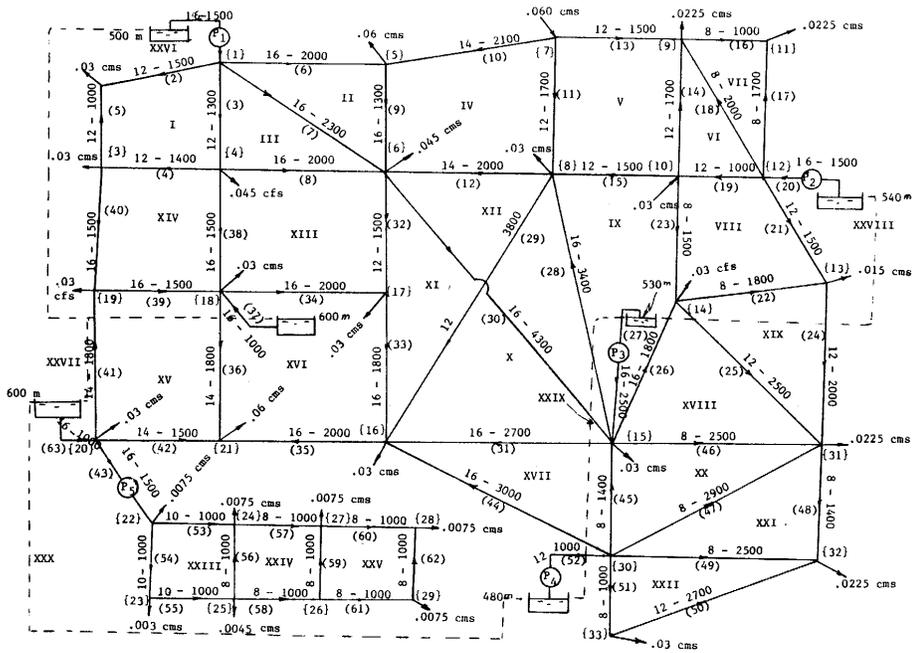
Junction No.	1	2	3	4	5	6	7	8	9	10
Elev. (m)	300	250	255	260	280	200	340	270	330	360

Junction No.	11	12	13	14	15	16	17	18	19	20
Elev. (m)	260	420	270	240	260	260	260	280	270	300

Junction No.	21	22	23	24	25	26	27	28	29	30
Elev. (m)	250	260	200	160	160	70	70	40	20	260

Junction No.	31	32	33
Elev. (m)	200	-50	-60

Verification Case 68 Problem Statement



LINEAR THEORY METHOD 107

108 PIPE NETWORK ANALYSES

Answer

Pipe No.	Flow Rate (m ³ /s)	Head Loss (m)	Pipe No.	Flow Rate (m ³ /s)	Head Loss (m)
1	0.0869	201.1	33	0.0046	0.80
2	0.0208	53.6	34	0.0274	27.6
3	0.0132	19.1	35	0.0094	3.5
4	0.0153	27.4	36	0.0219	31.8
5	0.0092	7.16	37	0.1306	297.8
6	0.0274	27.5	38	-0.0214	12.7
7	0.0254	27.3	39	-0.0299	24.4
8	0.0148	8.3	40	-0.0238	15.7
9	-0.0027	0.22	41	0.0239	37.8
10	0.0299	68.4	42	0.0287	45.2
11	0.0212	63.1	43	0.0450	54.6
12	0.0081	5.11	44	0.0139	11.1
13	0.0089	10.1	45	-0.0002	0.11
14	0.0164	38.1	46	0.0056	55.0
15	0.0273	91.0	47	0.0051	54.9
16	0.0107	78.9	48	0.0167	267.5
17	0.0118	165.5	49	0.0137	322.4
18	0.0078	86.5	50	-0.0078	14.3
19	0.0243	48.4	51	0.0222	336.7
20	0.0791	166.9	52	0.0547	240.7
21	0.0351	150.0	53	0.0210	94.2
22	-0.0035	16.3	54	0.0165	58.1
23	0.0106	117.7	55	0.0135	39.1
24	0.0166	45.8	56	-0.0020	2.97
25	0.0119	29.7	57	0.0116	93.0
26	0.0277	25.3	58	0.0109	82.9
27	0.0805	287.6	59	0.0031	7.07
28	-0.0043	1.37	60	0.0072	36.2
29	0.0061	12.52	61	0.0078	43.2
30	-0.0065	3.74	62	0.0003	0.12
31	0.0148	11.16	63	0.1276	284.5
32	0.0071	6.62			

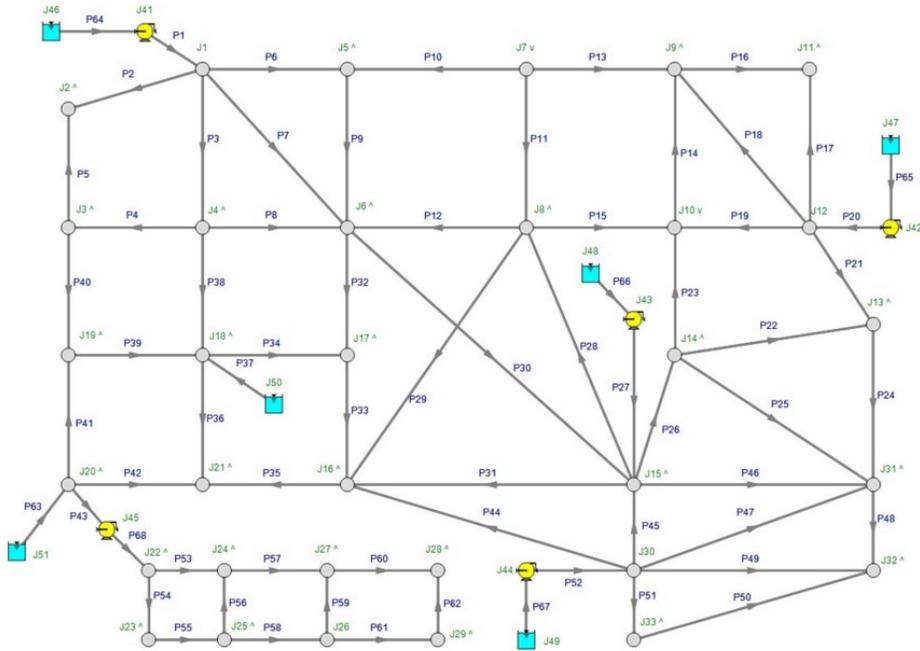
LINEAR THEORY METHOD 109

Junction	HGL (m)	Head (m)	Pressure (N/cm ²)
1	308.6	8.6	8.43
2	255.0	5.0	4.92
3	262.2	7.2	7.03
4	289.6	29.6	29.0
5	281.1	1.12	1.09
6	281.3	1.34	1.31
7	349.5	9.5	9.31
8	286.4	16.4	16.1
9	339.4	3.4	9.19
10	377.5	17.5	17.1
11	260.4	0.45	0.44
12	425.9	5.93	5.81
13	275.9	5.88	5.76
14	259.6	15.6	15.3
15	284.9	24.9	24.4
16	273.8	13.8	13.5
17	274.6	14.6	14.3
18	302.2	22.2	21.7
19	277.7	7.7	7.56
20	315.5	15.5	15.2
21	270.3	20.3	19.9
22	268.4	8.4	8.25
23	210.4	10.4	10.2
24	174.2	14.2	14.0
25	171.3	11.3	11.0
26	88.3	18.3	17.9
27	81.3	11.3	11.0
28	45.1	5.1	5.0
29	45.2	25.2	24.7
30	284.8	24.8	24.4
31	230.1	30.1	29.5
32	-37.5	12.5	12.2
33	-51.8	8.2	8.0

11. Solve the 8-pipe network shown in the accompanying sketch which is supplied by two reservoirs with water surfaces at 180 and 200 ft, respectively. The PRV is located 450 ft downstream from the upstream junction of pipe 5 and has a pressure setting which maintains the elevation of the HGL at 50 ft downstream from the valve.

View Verification Case 68 Model

[Verification Case 68](#)



Verification Case 69

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom

TITLE: FthVerify69.fth

REFERENCE: Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 102-105

FLUID: Water

ASSUMPTIONS: Assume water at 70 deg. F.

RESULTS:

Pipe Flow Rate (ft3/sec)	1	2	3	4	5	6	7	8
Jeppson	11.61	3.18	2.18	1.5	2.22	-1.19	0.3	-0.68
AFT Fathom	11.506	3.175	2.175	1.498	2.201	-1.123	0.380	-0.670

Pipe Flow Rate (ft3/sec)	9	10	11	12	13	14	15	16
Jeppson	0.49	-1.43	2.63	2.06	10	0.93	1.49	1.89
AFT Fathom	0.570	-1.400	2.520	1.987	9.937	1.074	1.535	1.821

Pipe Flow Rate (ft3/sec)	17	18	19	20	21	22	23	24
Jeppson	1.54	1.47	2.75	2.71	-0.26	-1.45	-1.77	-1.71
AFT Fathom	1.322	1.396	2.480	2.461	-0.189	-1.271	-1.558	-1.543

Pipe Flow Rate (ft3/sec)	25	26	27	28	29	30	31	32
Jeppson	0.58	2.09	-2.1	0.07	1.18	2.73	1.14	-0.92
AFT Fathom	0.551	1.973	1.980	-0.065	1.055	2.561	1.089	-0.754

Pipe Flow Rate (ft3/sec)	33	34	35	36	37	38	39	40
Jeppson	4.72	3.51	0.71	-1.95	-2.72	0.88	-0.43	-3.72
AFT Fathom	4.632	3.298	0.642	-1.748	-2.439	0.815	-0.182	-3.344

Pipe Flow Rate (ft3/sec)	41	42	43	44	45	46	47	48
Jeppson	1.05	-1	-0.86	-0.5	-0.4	1.45	1.43	0.35
AFT Fathom	0.867	-0.893	-0.514	0.505	-0.444	1.307	1.307	0.361

Verification Case 69

Pipe Flow Rate (ft3/sec)	49	50	51	52	53	54	55	56
Jeppson	2.55	1.45	4.25	9.03	1.24	-3.02	3.27	-1.21
AFT Fathom	2.462	1.387	4.125	8.950	1.582	-2.890	3.472	-1.600

Pipe Flow Rate (ft3/sec)	57	58	59	60	61	62	63	64
Jeppson	-0.22	-1.22	0.92	-0.8	1.8	6.25	-5.14	2.8
AFT Fathom	-0.487	-1.141	0.903	-0.738	1.738	6.044	-3.950	2.667

Pipe Flow Rate (ft3/sec)	65
Jeppson	7
AFT Fathom	6.263

Pipe Head Loss (feet)	1	2	3	4	5	6	7	8
Jeppson	82.15	14.2	9.23	23.43	26.31	2.87	0.18	3.6
AFT Fathom	*80.43	14.15	9.18	23.32	25.86	** -2.54	0.27	** -3.51

Pipe Head Loss (feet)	9	10	11	12	13	14	15	16
Jeppson	0.73	3.42	7.68	11.1	25.67	1.89	5.33	11.07
AFT Fathom	0.97	** -3.24	7.01	10.25	*26.74	2.48	5.58	10.27

Pipe Head Loss (feet)	17	18	19	20	21	22	23	24
Jeppson	3.78	3.44	7.22	7.04	0.18	4.21	4.03	5.74
AFT Fathom	2.78	3.09	5.87	5.78	** -0.09	** -3.22	** -3.13	** -4.69

Pipe Head Loss (feet)	25	26	27	28	29	30	31	32
Jeppson	1.71	19.84	18.13	0.02	3.26	21.37	20.22	1.88
AFT Fathom	1.56	17.70	16.14	** -0.02	2.62	18.78	18.49	** -1.26

Verification Case 69

Pipe Head Loss (feet)	33	34	35	36	37	38	39	40
Jeppson	28.18	28.34	8.01	22.3	14.28	0.85	0.13	21.31
AFT Fathom	27.12	24.95	6.46	** -17.95	** -11.49	0.73	** -0.03	** -17.20

Pipe Head Loss (feet)	41	42	43	44	45	46	47	48
Jeppson	2.4	4.25	1.63	90	0.9	4.75	4.87	0.79
AFT Fathom	1.64	** -3.37	** -0.61	1.70	** -1.06	3.85	4.08	0.84

Pipe Head Loss (feet)	49	50	51	52	53	54	55	56
Jeppson	17.59	4.21	13.38	26.61	3.32	24.63	18.65	1.55
AFT Fathom	16.38	3.82	12.57	*26.06	5.25	** -22.45	*20.90	** -0.26

Pipe Head Loss (feet)	57	58	59	60	61	62	63	64
Jeppson	0.08	3.78	1.17	1.15	4.23	12.87	19.47	16.8
AFT Fathom	** -0.343	** -3.312	1.111	** -0.981	3.946	*12.006	** -29.681	15.549

Pipe Head Loss (feet)	65
Jeppson	23.88
AFT Fathom	23.043

Node EGL (feet)	1	2	3	4	5	6	7	8
Jeppson	390.2	376	366.7	363.9	366.6	363.1	374.2	372.3
AFT Fathom	393.20	379.10	369.90	367.40	369.60	366.40	376.70	374.20

Node EGL (feet)	9	10	11	12	13	14	15	16
Jeppson	376.1	369.1	368.9	363.2	345	341.8	362	333.8
AFT Fathom	377.00	371.20	371.10	366.40	350.20	347.60	366.10	341.20

Node EGL (feet)	17	18	19	20	21	22	23	24
Jeppson	340.9	343.3	348.1	364.9	351.5	347.3	342.4	340.8
AFT Fathom	346.90	348.50	352.40	367.90	355.40	351.60	347.50	346.90

Node EGL (feet)	25	26	27	28	29
Jeppson	319.5	316.1	342.3	346.1	347.3
AFT Fathom	329.70	324.40	347.10	350.50	351.40

* AFT Fathom results combine two pipes, as discussed below

** Note that AFT Fathom represents head loss on pipes with reverse flow as a negative. Jeppson represents it as positive regardless of the direction.

DISCUSSION:

Jeppson's method of applying pump data is to lump it into a pipe, whereas AFT Fathom's method is to place pumps at boundaries between pipes. Pumps are therefore a specific node (or junction) in AFT Fathom. To accommodate Jeppson's method, the pipe which contains the pump is split into two equivalent pipes in AFT Fathom. Where the split is made will have no impact on the results.

Because there are five pumps in the example, there are five additional pipes in the AFT Fathom model. AFT Fathom pipes 1 and 71 together represent Jeppson pipe 1. Similarly, AFT Fathom pipes 13 and 72 represent Jeppson pipe 13, AFT Fathom pipes 52 and 73 represent Jeppson pipe 52, AFT Fathom pipes 55 and 75 represent Jeppson pipe 55, and AFT Fathom pipes 62 and 74 represent Jeppson pipe 62.

Jeppson presents results in terms of HGL. However, Jeppson's method assumes EGL and HGL are essentially the same because of minimal velocity. Therefore, Jeppson results are presented in the results shown above as EGL.

Results differ slightly between AFT Fathom and Jeppson for a few reasons. First, Jeppson represents pump curves differently than AFT Fathom. Jeppson typically uses an exponential formula (see page 82), while AFT Fathom uses a polynomial based on a least squares curve fit. Second, the head loss formula used by Jeppson differs from AFT Fathom. Jeppson's formula is more common to the water industry, and assumes the head loss is proportional to flow rate to some power near but less than 2. AFT Fathom assumes it always proportional to flow rate to the power of 2. These differences affect the results to some degree. Third, this system is highly networked which may cause some individual pipes, especially those with lower flow rates, to differ quite a bit from AFT Fathom. Looking at the system as a whole, the agreement is quite good between Jeppson and AFT Fathom.

Slight differences in property and calculation constants that were assumed, as well as potential differences from Jeppson's solution tolerances, which are not known, may also contribute to differences in the solution results. Examples are the specific value of water density and gravitational constant.

[List of All Verification Models](#)

Verification Case 69 Problem Statement

[Verification Case 69](#)

Roland Jeppson, Analysis of Flow in Pipe Networks, 1976, Publisher Ann Arbor Science, Page 102-105

[Jeppson Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

102 PIPE NETWORK ANALYSES

This solution took seven iterations to meet the error requirement that the sum of flow rate changes be less than 0.01 and required 11.7 seconds of execution time on a UNIVAC 1108 computer.

9. Solve the 65 pipe network below for the flow rates in all pipes, the head losses in all pipes, the elevation of the HGL at all junctions, and the pressure at all junctions. All pipes have $e = 0.0102$ inch. Three values of flow rate and head loss for the five pumps in the network are:

Pump No.	Point 1		Point 2		Point 3	
	Q_p	h_p	Q_p	h_p	Q_p	h_p
1	8.	200.	11.	180.	16.	80.
2	5.	180.	7.5	150.	10.	50.
3	4.	200.	6.	180.	10.	80.
4	4.	250.	6.	210.	8.	150.
5	2.	50.	4.	40.	6.	20.

The elevations of the junctions are:

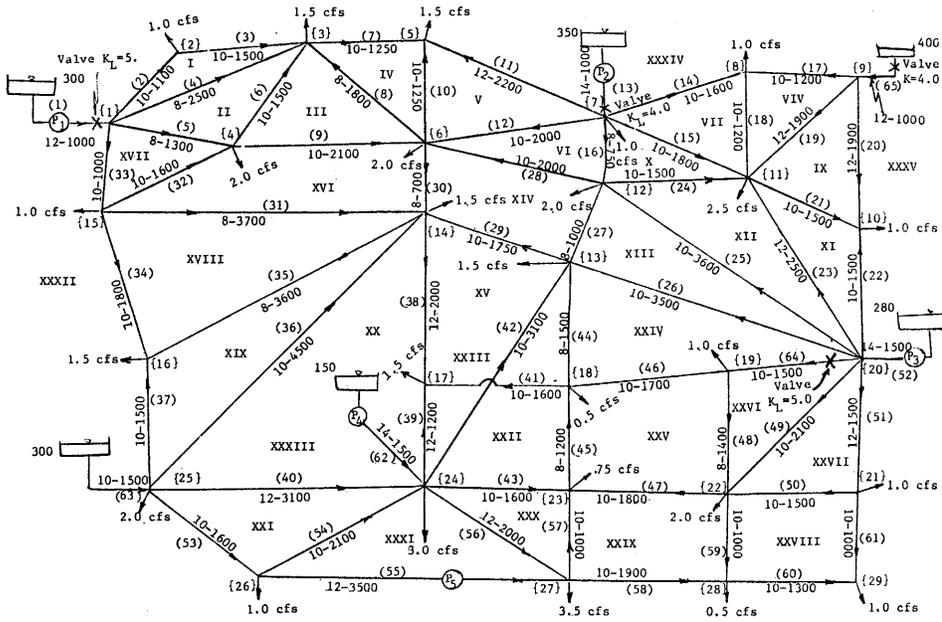
Junction No.	Elev. ft	Junction No.	Elev. ft	Junction No.	Elev. ft
1	150	11	190	21	170
2	150	12	190	22	160
3	160	13	180	23	150
4	150	14	160	24	150
5	165	15	150	25	140
6	160	16	150	26	130
7	180	17	140	27	130
8	190	18	150	28	130
9	200	19	170	29	125
10	180	20	180		

Solution:

Pipe No.	1	2	3	4	5	6	7
Q (cfs)	11.61	3.18	2.18	1.50	2.22	-1.19	0.30
h_f (ft)	82.15	14.20	9.23	23.43	26.31	2.87	0.18

Pipe No.	8	9	10	11	12	13	14
Q (cfs)	-0.68	0.49	-1.43	2.63	2.06	10.00	0.93
h_f (ft)	3.60	0.73	3.42	7.68	11.10	25.67	1.89

Verification Case 69 Problem Statement



LINEAR THEORY METHOD
103

Verification Case 69 Problem Statement

104 PIPE NETWORK ANALYSES

Pipe No.	15	16	17	18	19	20	21
Q (cfs)	1.49	1.89	1.54	1.47	2.75	2.71	-0.26
h_f (ft)	5.33	11.07	3.78	3.44	7.22	7.04	0.18

Pipe No.	22	23	24	25	26	27	28
Q (cfs)	-1.45	-1.77	-1.71	0.58	2.09	-2.10	0.07
h_f (ft)	4.21	4.03	5.74	1.71	19.84	18.13	0.02

Pipe No.	29	30	31	32	33	34	35
Q (cfs)	1.18	2.73	1.14	-0.92	4.72	3.51	0.71
h_f (ft)	3.26	21.37	20.22	1.88	28.18	28.34	8.01

Pipe No.	36	37	38	39	40	41	42
Q (cfs)	-1.95	-2.72	0.88	-0.43	-3.72	1.05	-1.00
h_f (ft)	22.30	14.28	0.85	0.13	21.31	2.40	4.25

Pipe No.	43	44	45	46	47	48	49
Q (cfs)	-0.86	-0.50	-0.40	1.45	1.43	0.35	2.55
h_f (ft)	1.63	1.72	0.90	4.75	4.87	0.79	17.59

Pipe No.	50	51	52	53	54	55	56
Q (cfs)	1.45	4.25	9.03	1.24	-3.02	3.27	-1.21
h_f (ft)	4.21	13.38	26.61	3.32	24.63	18.65	1.55

Pipe No.	57	58	59	60	61	62	63
Q (cfs)	-0.22	-1.22	0.92	-0.80	1.80	6.25	-5.14
h_f (ft)	0.08	3.78	1.17	1.15	4.23	12.87	19.47

Pipe No.	64	65
Q (cfs)	2.80	7.00
h_f (ft)	16.80	23.88

LINEAR THEORY METHOD 105

Junction No.	1	2	3	4	5	6
HGL (ft)	390.2	376.0	366.7	363.9	366.6	363.1
Pressure (psi)	104.1	97.9	89.6	92.7	87.3	88.0

Junction No.	7	8	9	10	11	12
HGL (ft)	374.2	372.3	376.1	369.1	368.9	363.2
Pressure (psi)	84.2	79.0	76.3	81.9	77.5	75.0

Junction No.	13	14	15	16	17	18
HGL (ft)	345.0	341.8	362.0	333.8	340.9	343.3
Pressure (psi)	71.5	78.8	91.9	79.6	87.1	83.8

Junction No.	19	20	21	22	23	24
HGL (ft)	348.1	364.9	351.5	347.3	342.4	340.8
Pressure (psi)	77.2	80.1	78.65	81.2	83.4	82.7

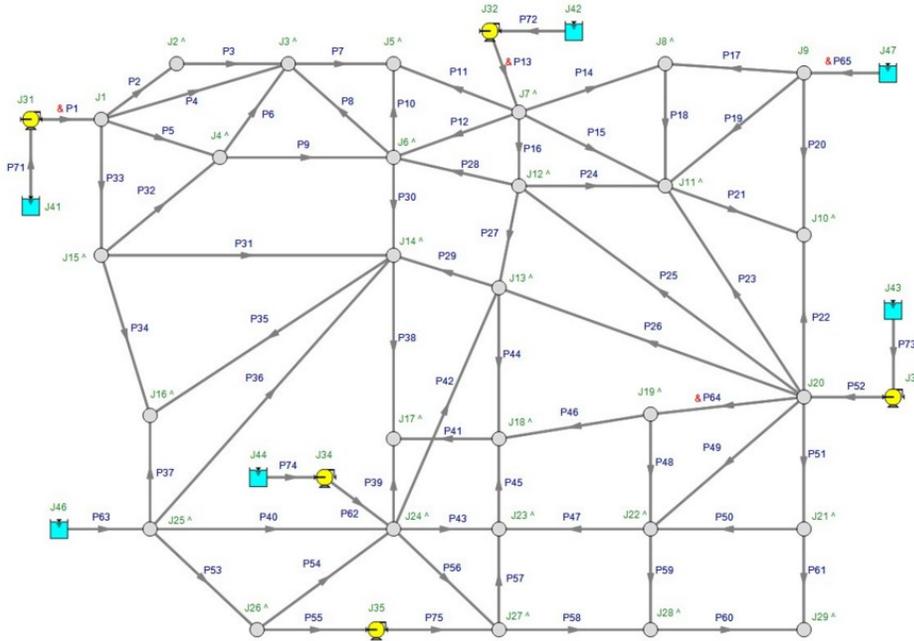
Junction No.	25	26	27	28	29
HGL (ft)	319.5	316.1	342.3	346.1	347.3
Pressure (psi)	77.8	80.7	92.0	93.6	96.3

Using the linear theory program described in Appendix C, the solution was obtained in six iterations with a sum of changes in flow rate equal to 0.019. The solution requires 23.3 seconds of execution time on a UNIVAC 1108.

- Determine the flow rates and head losses of each of the 63 pipes of the network shown below. The diameter of each pipe is given in centimeters and the length of each pipe is given in meters. Thus pipe number 1 is 16 cm in diameter and has a length of 1500 m. All 63 pipes of the network have a wall roughness of 0.026 cm. The network is supplied by 2 reservoirs without pumps and 4 pumps obtaining their water from the reservoirs as shown in the sketch. A booster pump exists in pipe 43 also. The water surface elevations are as given on the sketch in meters and the pump characteristic curves are defined by the data for three points as given below.

View Verification Case 69 Model

[Verification Case 69](#)



Verification Case 70

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom (with SSL Module)

TITLE: FthVerify70.fth

REFERENCE: Wilson, Addie, Sellgren & Clift, Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer, Page 145-147, Case Study 6.1

FLUID: Water/Sand

ASSUMPTIONS: Water temperature is assumed to be 4 C to match the specified fluid specific gravity of 1. The pipe length and inlet pressure were not specified in the problem statement. They were assumed to be 100 m and 1 MPa, respectively. The choice of pipe length and inlet pressure does not affect the resulting calculations. It was also assumed that $Vt/Vts(Xi) = 0.55$ for coarse sand.

RESULTS:

Wilson, Addie, Sellgren & Clift Results:

Case	Velocity (meters/sec)	Cv (Decimal)	Im (m/m)	SEC (km)	(kW-hr/mton-)
Cvd 0.221	5.70	0.221	0.0609	0.289	
Cvd 0.2	6.30	0.200	0.0612	0.315	
Cvd 0.183	6.90	0.183	0.0636	0.358	

AFT Fathom Results:

Case	Velocity (meters/sec)	Cv (Decimal)	Im (m/m)	SEC (km)	(kW-hr/mton-)
Cvd 0.221	5.72	0.221	0.0613	0.285	
Cvd 0.2	6.32	0.200	0.0609	0.313	
Cvd 0.183	6.91	0.183	0.0633	0.355	

DISCUSSION:

There is close agreement between the results for AFT Fathom and the results for Wilson, Addie, Sellgren and Clift.

[List of All Verification Models](#)

Verification Case 70 Problem Statement

[Verification Case 70](#)

Wilson, Addie, Sellgren & Clift, Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer, Pages 145-146, Case Study 6.1 a,b

[Wilson, Addie, Sellgren & Clift's Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

6. Heterogeneous Slurry Flow in Horizontal Pipes

145

substantially exceeds 0.4, the solids are almost uniformly distributed over the pipe section. This observation is consistent with the close particle spacing at high concentration which will reduce the effectiveness of turbulent suspension, so that the particles' submerged weight must be carried by granular contact. Further increases in C_v lead to larger hydraulic gradients and a pronounced rise in specific energy consumption.

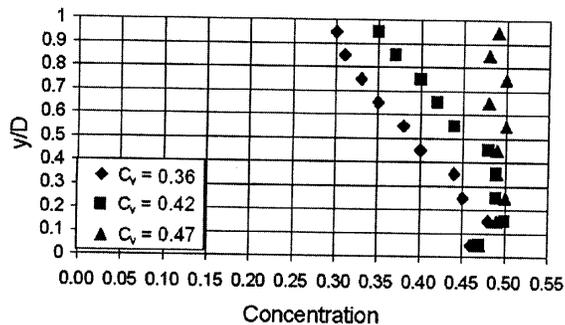


Figure 6.9. Concentration profiles for 180 μm sand tested by Gillies & Shook (1996)

The next step in describing slurry flows will be to discuss complex slurries – those with features such as very broad particle grading or non-Newtonian carrier fluids. Complex slurries are the subject of Chapter 7.

6.8 Case Studies

Case Study 6.1 - Effect of Particle Size and Grading on Sand Transport

In Case Study 5.1 a preliminary pipe selection was made for a system to transport 4000 tonnes/hour of coarse sand. One of the pipe sizes selected was $D = 0.65$ m, for which the combination $V_m = 6.30$ m/s and $C_{vd} = 0.20$ gives the required solids flow rate.

In that case study the mean size and grading of the particles was not specified; and thus the hydraulic gradient and the specific energy consumption could not be calculated. The present case study will investigate these points.

The first material to be considered is a clean sand (negligible fraction smaller than 40 μm) with $S_s = 2.65$, $d_{50} = 0.70$ mm and $d_{85} = 1.00$ mm. We

will consider an aqueous slurry ($S_r = 1.00$) in the pipe with $D = 0.65$ m. The conditions found in Case Study 5.1 ($C_{vd} = 0.20$, $V_m = 6.30$ m/s) will be investigated initially, and the first requirement will be:

(a) Find the hydraulic gradient i_m and the specific energy consumption for this case. For these particles the associated velocity w can be read directly from Fig. 6.5, giving 0.127 m/s for the 0.70 mm particle and 0.148 m/s for the 1.00 mm one. When multiplied by $\cosh(60d/D)$, these become 0.1273 m/s and 0.1486 m/s, giving a ratio of 1.17. From Eq. 6.8, the associated value of σ_s is 0.069, and when this is substituted into Eq. 6.9, M is given as 1.79. As this is larger than 1.7, $M = 1.7$ will be used. (This result is not surprising, since d_{85}/d_{50} is less than 1.5, indicating a narrow particle grading).

With M evaluated, we can turn to V_{50} , equal to $w \sqrt{\frac{8}{f_w}} \cosh(60d/D)$, based on d_{50} . For $f_w = 0.012$ (found from the

Stanton-Moody diagram for commercial steel pipe of this size and velocity range), the resulting value of V_{50} is 3.28 m/s. If the simplified approach of Section 6.5 had been used, M would remain at 1.7 and V_{50} would be 3.47 m/s. On the basis of mixture velocity $V_m = 6.30$ m/s, and $(S_m - S_f) = (1.65) 0.20 = 0.330$, Eq. 6.4 gives the solids effect ($i_m - i_f$) as $(0.330)(0.22)[1.921]^{-1.7}$ or 0.0239 (m water/m pipe).

With no fines, the fluid is water and $i_f = i_w$, given by $0.012 (6.3)^2 / (19.62)(0.65)$, or 0.0373. The value of i_m is the sum of i_w and the solids effect or 0.0612 (m water/m pipe).

The specific energy consumption is proportional to $i_m/S_s C_{vd}$, which is 0.115, giving SEC of 0.315 kWh/tonne-km.

(b) Next, it is of interest to vary some of the quantities and as input, to find the effect on i_m and SEC. The initial case to be investigated is a variation of V_m only, with C_{vd} kept constant. The second case involves adjusting C_{vd} as well as V_m so that the solids throughput Q_s is maintained at a constant value.

The effect of varying V_m alone could be obtained by differentiation, but it is probably clearer simply to change V_m by about 10% in each direction, repeating the calculations for, say, $V_m = 6.9$ m/s and 5.7 m/s. As i_w varies with V_m^2 (ignoring any changes in f_w) and $(i_m - i_w)$ varies with $V_m^{1.7}$, the calculations are straightforward. For $V_m = 6.9$ m/s, $i_m = 0.0653$ and SEC = 0.336

kWh/tonne-km. For $V_m = 5.7$ m/s, $i_m = 0.0589$ and $SEC = 0.304$ kWh/tonne-km.

If the same values of V_m are employed, but C_{vd} is adjusted to maintain Q_s , the values shown in Table 6.1 are obtained.

Table 6.1.

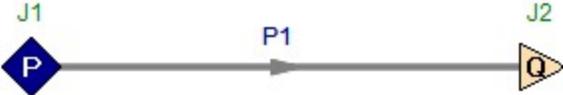
V_m (m/s)	C_{vd}	i_m	SEC (kWh/tonne-km)
5.70	0.221	0.0619	0.289
6.30	0.200	0.0612	0.315
6.90	0.183	0.0636	0.358

As with the figures for the constant- C_{vd} case, this table indicates a significant increase in SEC when V_m is increased to 6.90 m/s. As before, the specific energy consumption is diminished when V_m is decreased to 5.70 m/s, but once again, the change is smaller for a decrease in velocity. Thus it would appear that the lower velocity of 5.7 m/s could be attractive. On the other hand, the lower velocity is less than 20 percent above the deposition limit ($V_{sm} = 4.84$ m/s by Eq. 5.11, see Case Study 5.1), and considerations of system stability may be involved. In order to resolve these questions, it will be necessary to know the pump characteristic as well as the system characteristic. Pump characteristics will be discussed in Chapter 9, and the intersection of pump and system characteristics in Chapter 13.

(c) It is also of interest to find the effect on i_m of changes in particle grading. As an initial example, suppose that the coarse sand specified above is replaced by a fine, clean, narrow-graded sand with $d_{50} = 0.20$ mm. From Fig. 6.5, w for this sand is 0.080 m/s. In the pipe of 0.65 m diameter the resulting value of V_{50} from Eq. 6.2 is 2.08 m/s. For $V_m = 6.3$ m/s, and $C_{vd} = 0.20$, the solids effect as calculated from Eq. 6.4 is 0.0110, giving $i_m = 0.0483$ m water/m pipe. (If the simplified approach had been used the value of i_m would be 0.0498 m water/m pipe). Clearly, i_m is considerably less than the value of 0.0612 found in part (a) for the coarse sand. It may be compared with the result of the 'equivalent fluid' model, by which i_m is estimated as the product of S_m and i_w . As $S_m = 1.330$, and $i_w = 0.0373$, the equivalent fluid model gives $i_m = 0.0496$, rather close to the result of the new model for this specific instance of fine-sand slurry flow.

View Verification Case 70 Model

[Verification Case 70](#)



Verification Case 71

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom (with SSL Module)

TITLE: FthVerify71.fth

REFERENCE: Wilson, Addie, Sellgren & Clift, Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer, Page 148-149, Case Study 6.2

FLUID: Water/Coal

ASSUMPTIONS: The water temperature is not specified. It was assumed to be 20 deg. C because the viscosity ratio in equation 6.11 in the reference is based on the viscosity at 20 deg. C. The pipe length and inlet pressure were not given. They were assumed to be 100 m and 5 bar, respectively. The choice of pipe length and inlet pressure does not affect the resulting calculations. The calculation was run once to determine the mass flow rate of 1857 m-ton/hr at the minimum I_m , then run again with the flow set to this value. In this way, the proper value of SEC was determined for the calculation. Also, while AFT Fathom calculates the expected value of 1.94 m/s for the Settling Velocity, the example in the reference uses a value of 1.90 m/s, so this value was used for the calculations in AFT Fathom as well. Assume $M = 1.7$.

RESULTS:

	V50 (meters/sec)	Velocity at min I_m (meters/sec)	I_m (m/m)	SEC (kW-hr/mton-km)
Wilson, Addie, Sellgren & Clift	2.64	3.1	0.0313	0.244
AFT Fathom	2.65	3.1	0.0313	0.243

[List of All Verification Models](#)

Verification Case 71 Problem Statement

[Verification Case 71](#)

Wilson, Addie, Sellgren & Clift, Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer, Pages 148-149, Case Study 6.2

[Wilson, Addie, Sellgren & Clift's Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

148

Chapter 6

Consider next a coarse sand with $d_{50} = 0.70$ mm as before, but with a broader particle grading so that $d_{85} = 1.50$ mm. For the latter size, $w = 0.18$ m/s, and for the 0.65 m pipe $\cosh(60d/D)$ is 1.01. Substitution into Eq. 6.8 shows that the value of σ_s for this material is 0.145. When this is substituted into Eq. 6.9, M is found to be 1.38. (The simplified approach gives $M = 1/\ln(1.50/0.70) = 1.31$.)

The value of V_{50} remains at 3.28 m/s and for $V_m = 6.30$ m/s and $C_{vd} = 0.20$, the solids effect becomes $(0.330)(0.22)[1.921]^{-1.38}$ or 0.0295. Adding i_w of 0.0373 gives $i_m = 0.0668$ m water/m pipe, which is about 9% higher than the value obtained previously for the narrow-graded particle distribution with the same median size. (A similar increase, about 11%, is obtained if the simplified approach is used throughout.)

Case Study 6.2 - Transport of Coal Slurry

The material for this study is coal, with $S_s = 1.4$, $\mu_s = 0.44$, $C_{vb} = 0.60$. The coal is clean with few fines, and $d_{50} = 2.0$ mm and $d_{85} = 2.8$ mm (as the ratio of d_{85}/d_{50} is less than 1.5, the particle grading is narrow, and M will be taken as 1.7).

The pipe to be used has $D = 0.440$ m and $f_w = 0.013$. Solids concentration C_{vd} can be taken initially as 0.25.

The minimum in the curve of constant C_{vd} is to be investigated, comparing the velocity at this point with V_{sm} and V_{50} , and finding the minimum specific energy consumption and how this is influenced by changes in C_{vd} .

(a) the limit of deposition will be determined first. For $d = 2.0$ mm and $D = 0.440$ m, Fig. 5.3 shows that V_{sm} would be 4.1 m/s for material with $S_s = 2.65$, and for $S_s = 1.4$, the right-hand side of that figure gives $V_{sm} = 1.90$ m/s. Equation 5.11 gives a value of 1.94 m/s, but the smaller number should be taken, i.e. $V_{sm} = 1.90$ m/s. (Note that for these 2.0 mm coal particles the nomographic chart shows a smaller value than Eq. 5.10; the reverse of the condition found in Case Study 5.1 for 0.7 mm sand).

(b) To obtain V_{50} it is sufficient to use the simplified approach of Eq. 6.11 which gives 2.64 m/s. For this narrow-graded material $M = 1.7$, and with $(S_s - 1)C_{vd} = 0.4(0.25) = 0.10$, the expression for solids effect (Eq. 6.4) becomes:

$$i_m - i_w = 0.022 \left(\frac{V_m}{2.64} \right)^{-1.7} = 0.115 V_m^{-1.7}$$

Also, i_w is $\frac{0.013V_m^2}{(19.62)(0.44)} = 0.00151V_m^2$. The sum of these two terms is i_m , and differentiation gives the following relation for the velocity at minimum i_m :

$$1.7(0.115)V_m^{2.7} = 2(0.00151)V_m$$

from which V_m equals 3.1 m/s. At this point i_m is calculated to be 0.0313 m water/m pipe and $i_m/S_s C_{vd}$ is 0.089, giving SEC = 0.244 kWh/tonne-km. (This is the quantity which must be provided by the pumps to the slurry, the energy which must be supplied to the pumps will, of course, be larger).

(c) In this case, the value of V_m at the minimum point is comfortably above the deposition limit of 1.9 m/s, so no problem with deposition should arise. However, stability problems would occur near the minimum, as shown in Chapter 13, and thus the operating velocity should be larger than 3.1 m/s. It is also worth noting that in the present study, the flow is definitely heterogeneous near the minimum point. At the minimum point V_m/V_{50} equals 1.17 and the stratification ratio, which is given by $0.5 (V_m/V_{50})^{1.7}$ equals 0.38, implying that about 38% of the submerged weight of solids is carried by intergranular contact, with fluid support accounting for the rest.

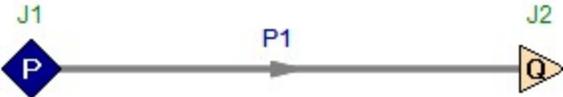
(d) Finally, it is of interest to consider the effect of a change in concentration on the minimum point. As shown in the discussion following Eq. 6.5, the velocity at minimum i_m varies *ceteris paribus* with $C_{vd}^{0.27}$. Thus, if C_{vd} increases from 0.25 to 0.30 the value of V_m at the minimum point will increase from 3.1 m/s to 3.25 m/s; not a large change. It can readily be calculated that the equivalent value of i_m is 0.0345 m water/m pipe. This gives SEC = 0.224 kWh/tonne-km, or about 92% of the value obtained previously for $C_{vd} = 0.25$.

REFERENCES

- Carstens, M.R. & Addie, G.R. (1981). A sand-water slurry experiment. *Jour. Hydr. Div. ASCE*, Vol. 107, No. HY4, p. 501-507.
- Clift, R., Wilson, K.C., Addie, G.R., & Carstens, M.R. (1982). A mechanistically-based method for scaling pipeline tests for settling slurries. *Proc. Hydrotransport 8*, BHRA Fluid Engineering, Cranfield, UK, pp. 91-101.

View Verification Case 71 Model

[Verification Case 71](#)



Verification Case 72

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom (with SSL Module)

TITLE: FthVerify72.fth

REFERENCE: Wilson, Addie, Sellgren & Clift, Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer, Page 243, Case Study 10.1

FLUID: Water/Sand

ASSUMPTIONS: Water temperature is assumed to be 4 C. M is assumed to be 1.7. Also, the system configuration is not given (i.e. pressures, pipe lengths, pipe diameters, etc) and were assumed. These assumptions do not affect the final results of the problem. Use simplified slurry calculations. Use ANSI/HI Standard 12.1-12.6-2005 calculations to determine the pump corrections.

RESULTS:

	AFT Fathom	Wilson, Addie, Sellgren & Clift
De-Rating Correction:	4.7%	4.7%

DISCUSSION:

The correction factor calculated in AFT Fathom was CH = 95.3%. This is related to the correction given in Wilson, Addie, Sellgren & Clift by subtracting CH from 100%.

[List of All Verification Models](#)

Verification Case 72 Problem Statement

Verification Case 72

Wilson, Addie, Sellgren & Clift, Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer, Pages 243, Case Study 10.1

[Wilson, Addie, Sellgren & Clift's Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

10. Effect of Solids on Pump Performance

243

concentrations, specifically with broad particle size distributions.

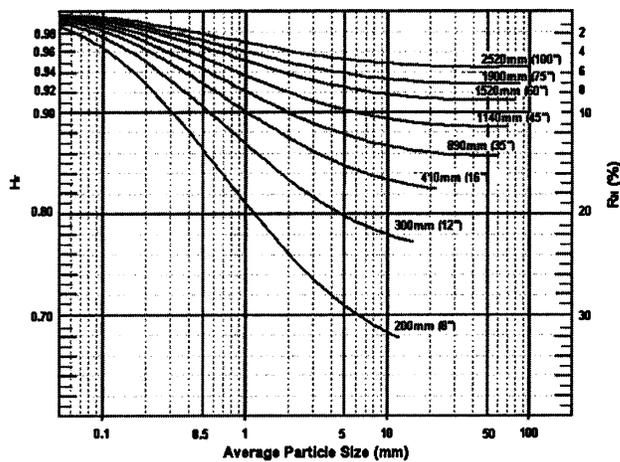


Figure 10.9. Generalised solids-effect diagram for pumps of various sizes (impeller diameters) based mainly on tests at The GIW Hydraulic Laboratory. For solids concentration by volume, $C_{vd} = 15\%$ with relative density of solids, $S_s = 2.65$ and a negligible amount of fine particles

Example 10.1

A sand slurry is to be pumped using a large pump with a 1320 mm impeller. Determine the reduction in head R_H if the solids concentration by volume $C_{vd} = 0.20$, the solids specific gravity $S_s = 2.65$ and the average particle size $d_{50} = 400 \mu\text{m}$.

It follows from the diagram that $R_H = 3.5\%$ for $S_s = 2.65$, $d_{50} = 400 \mu\text{m}$ and $C_{vd} = 15\%$. Correction for $C_{vd} = 20\%$ gives: $3.5 (20/15) = 4.7\%$.

Example 10.2

Determine R_H for a pump with an impeller diameter of 800 mm pumping an ore product ($S_s = 4.0$) with $d_{50} = 500 \mu\text{m}$, $X_h = 0.28$ and delivered volumetric concentration of 20%.

View Verification Case 72 Model

[Verification Case 72](#)



Verification Case 73

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom (with SSL Module)

TITLE: FthVerify73.fth

REFERENCE: Wilson, Addie, Sellgren & Clift, Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer, Page 243-244, Case Study 10.2

FLUID: Water/Ore

ASSUMPTIONS: Water temperature is assumed to be 4 deg. C. M is assumed to be 1.7. Also, the system configuration is not given (i.e. pressures, pipe lengths, pipe diameters, etc) and were assumed. These assumptions do not affect the final results of the problem. Use simplified slurry calculations. Use ANSI/HI Standard 12.1-12.6-2005 calculations to determine the pump corrections.

RESULTS:

	AFT Fathom	Wilson, Addie, Sellgren & Clift
De-Rating Correction:	6.4%	6.2%

DISCUSSION:

The correction factor calculated in Fathom was CH = 93.6%. This is related to the correction given in Wilson, Addie, Sellgren & Clift by subtracting CH from 100%.

[List of All Verification Models](#)

Verification Case 73 Problem Statement

Verification Case 73

Wilson, Addie, Sellgren & Clift, Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer, Pages 243-244, Case Study 10.2

[Wilson, Addie, Sellgren & Clift's Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

10. Effect of Solids on Pump Performance

243

concentrations, specifically with broad particle size distributions.

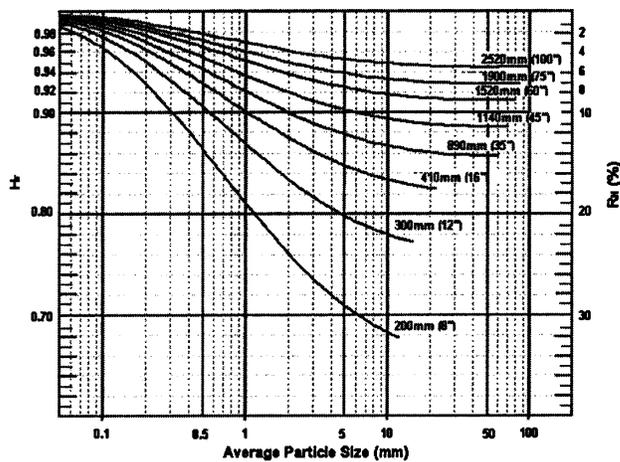


Figure 10.9. Generalised solids-effect diagram for pumps of various sizes (impeller diameters) based mainly on tests at The GIW Hydraulic Laboratory. For solids concentration by volume, $C_{vd} = 15\%$ with relative density of solids, $S_s = 2.65$ and a negligible amount of fine particles

Example 10.1

A sand slurry is to be pumped using a large pump with a 1320 mm impeller. Determine the reduction in head R_H if the solids concentration by volume $C_{vd} = 0.20$, the solids specific gravity $S_s = 2.65$ and the average particle size $d_{50} = 400 \mu\text{m}$.

It follows from the diagram that $R_H = 3.5\%$ for $S_s = 2.65$, $d_{50} = 400 \mu\text{m}$ and $C_{vd} = 15\%$. Correction for $C_{vd} = 20\%$ gives: $3.5 (20/15) = 4.7\%$.

Example 10.2

Determine R_H for a pump with an impeller diameter of 800 mm pumping an ore product ($S_s = 4.0$) with $d_{50} = 500 \mu\text{m}$, $X_h = 0.28$ and delivered volumetric concentration of 20%.

It follows from the diagram that $R_H = 6\%$ for $d_{50} = 500 \mu\text{m}$ for $S_s = 2.65$ and $X_H = 0$. Correction factor for $C_{vd} = 20\%$, $S_s = 4$ and $X_h = 0.28$ are; respectively:

$$\frac{20}{15} = 1.33$$

$$\left[\frac{4-1}{1.65} \right]^{0.65} = 1.475$$

$$(1 - 0.28)^2 = 0.52$$

R_H is then $6 \cdot 1.33 \cdot 1.47 \cdot 0.52 = 6.2\%$.

Example 10.3

A smaller pump is to be used to pump a slurry of the same solids as in Example 10.1, but at a volumetric concentration of 10%. The pump has an impeller diameter of 380 mm.

Following Example 10.1 with a pump impeller diameter 380 mm then the diagram gives $R_H = 7\%$. Correction for $C_{vd} = 10\%$ means that $R_H = 7 \cdot (10/15) = 4.7\%$, say 5%. Assuming that R_η equal to R_H , it is also 5%. The head required for this application is 45m of slurry at a flow rater of $0.0625 \text{ m}^3/\text{s}$. The pump has a 100mm discharge branch, 150mm suction branch and an impeller diameter of 380 mm. Its clear-water performance characteristics are shown on Figure 10.10. The running speed and power requirement for pumping this slurry are to be determined.

The head ratio H_r was defined in section 10.1 as $1 - R_H$, and for the present example this equals 0.95, i.e.

$$H_r = \frac{H_m}{H_w} = 0.95.$$

In order to be able to produce the required 45m head of slurry, the pump must be capable of producing a head of water, H_w , given by

$$H_w = \frac{45.0}{0.95} = 47.4\text{m of water}$$

View Verification Case 73 Model

[Verification Case 73](#)



Verification Case 74

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom (with SSL Module)

TITLE: FthVerify74.fth

REFERENCE: Wilson, Addie, Sellgren & Clift, Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer, Page 244-246, Case Study 10.3

FLUID: Water/Sand

ASSUMPTIONS: Water temperature is assumed to be 4 deg. C. M is assumed to be 1.7. The pipe length was assumed to be 144.5 m to achieve the specified flow of 62.5 l/s. The inlet and outlet pressures were assumed. These pressures do not affect the final results of the problem. Use simplified slurry calculations. Use ANSI/HI Standard 12.1-12.6-2005 calculations to determine the pump corrections.

RESULTS:

	AFT Fathom	Wilson, Addie, Sellgren & Clift
Corrected Efficiency	71.85%	71.3%
Corrected Power	48.51 kW	45.1 kW

DISCUSSION:

The answers in this case are close, but not exact. This is primarily due to some assumptions and rounding done in the example in the reference. For example, the flow rate of 62.5 l/s selected in the book assumed a speed of 1500 rpm. Examining figure 10.10, one can see that this is an approximation, so a slight difference in the pump head curve is introduced in this way. Also, the correction factor was rounded off from 4.7% to 5.0%. Based on these approximations, the results from AFT Fathom match very closely.

[List of All Verification Models](#)

Verification Case 74 Problem Statement

[Verification Case 74](#)

Wilson, Addie, Sellgren & Clift, Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer, Pages 244-246, Case Study 10.3

[Wilson, Addie, Sellgren & Clift's Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

244

Chapter 10

It follows from the diagram that $R_H = 6\%$ for $d_{50} = 500 \mu\text{m}$ for $S_s = 2.65$ and $X_H = 0$. Correction factor for $C_{vd} = 20\%$, $S_s = 4$ and $X_h = 0.28$ are, respectively:

$$\frac{20}{15} = 1.33$$

$$\left[\frac{4-1}{1.65} \right]^{0.65} = 1.475$$

$$(1 - 0.28)^2 = 0.52$$

$$R_H \text{ is then } 6 \cdot 1.33 \cdot 1.47 \cdot 0.52 = 6.2\% .$$

Example 10.3

A smaller pump is to be used to pump a slurry of the same solids as in Example 10.1, but at a volumetric concentration of 10%. The pump has an impeller diameter of 380 mm.

Following Example 10.1 with a pump impeller diameter 380 mm then the diagram gives $R_H = 7\%$. Correction for $C_{vd} = 10\%$ means that $R_H = 7 \cdot (10/15) = 4.7\%$, say 5%. Assuming that R_{η} equal to R_H , it is also 5%. The head required for this application is 45m of slurry at a flow rate of $0.0625 \text{ m}^3/\text{s}$. The pump has a 100mm discharge branch, 150mm suction branch and an impeller diameter of 380 mm. Its clear-water performance characteristics are shown on Figure 10.10. The running speed and power requirement for pumping this slurry are to be determined.

The head ratio H_r was defined in section 10.1 as $1 - R_H$, and for the present example this equals 0.95, i.e.

$$H_r = \frac{H_m}{H_w} = 0.95.$$

In order to be able to produce the required 45m head of slurry, the pump must be capable of producing a head of water, H_w , given by

$$H_w = \frac{45.0}{0.95} = 47.4\text{m of water}$$

For this water head, and the discharge of 62.5 L/s, the pump characteristics shown on Fig. 10.10 indicate that the pump must run at 1500 rpm, and that the clear water efficiency η_w will be 75%.

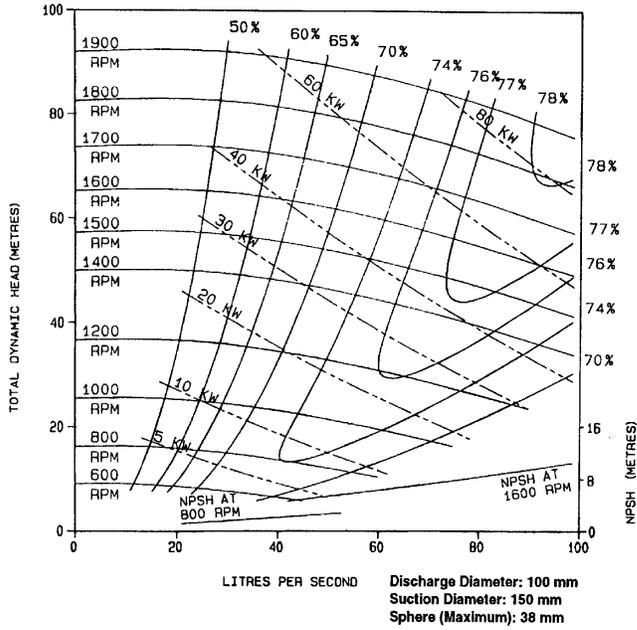


Figure 10.10. Pump characteristic curves for Example 10.4

With R_η assumed to be equal to R_{H1} , then

$$\eta_r = 1 - R_\eta = 0.95 = \frac{\eta_m}{\eta_w}$$

Thus

$$\eta_m = \eta, \eta_w = 0.95 (0.75) = 0.713 \text{ or } 71.3$$

The slurry specific gravity, S_m , is $1 + 0.10 (2.65 - 1)$ or 1.165, and by Eq. 10.1 P_m equals $S_m P_w$. For the calculated values of η_m and other quantities the power requirement is found to be

$$P_m = \frac{1.165(9.81)(45.0)(0.0625)}{0.713} = 45.1 \text{ kW}$$

REFERENCES

- Addie, G. & Hammer J. (1993): Pipeline head loss of a settling slurry at concentrations up to 49 % by volume, *Proc. 14th Int. Conference on the Hydraulic Transport of Solids in Pipe*, Brugge, Belgium, 28-30 September.
- Addie G. & Sellgren A. (2001): Application and construction of centrifugal solids handling pumps. Subsection 9.16.1. In Karassik I. J. et al. (2001): *Pump handbook*, Third ed. McGraw-Hill.
- Burgess, K.E., & Reizes, J.A. (1976). The effect of sizing, specific gravity and concentration on the performance of centrifugal slurry pumps. *Proc. Inst. Mech. Eng.*, UK, Vol. 190, No. 36, pp. 391-399.
- Engin & Gur (2003) Comparative evaluation of some existing correlations to predict head degradation of centrifugal slurry pumps. *ASME J. of Fluids Eng.* 125, January, pp149-157
- Gandhi B. K., Singh S. N. & Seshadri, V. (2001) Performance characteristics of centrifugal slurry pumps, *ASME J. Fluid Eng.* 123, pp 271-280
- Herbisch J.B. (1975) *Coastal and Deep Ocean Dredging*, Gulf Publishing., Houston TX, U.S.A.
- Holzenberger, K. (1980). Betriebsverhalten von Kreiselpumpen beim hydraulischen Feststofftransport. *VDI-Berichte* Nr. 371, pp. 59-66.
- Hydraulic Institute Standards for Centrifugal, Rotary and Reciprocating Pumps*, (2004) Hydraulic Institute, Cleveland, Ohio, Xth Ed.
- Kazim K.A., Maiti B. & Chand P. (1997) A correlation to predict the performance characteristics of centrifugal pumps handling slurries. *Proc. Instn. Mech. Engrs.*, 21A, pp 147-157
- McElvain, R.E. (1974). High pressure pumping. *Skilings Mining Review*, Vol. 63 No. 4, pp. 1-14.
- Newitt, D.M., Richardson, J.F., Abbott, M. & Turtle, R.B. (1955). *Hydraulic conveying of*

View Verification Case 74 Model

[Verification Case 74](#)



Verification Case 75

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom (with SSL Module)

TITLE:

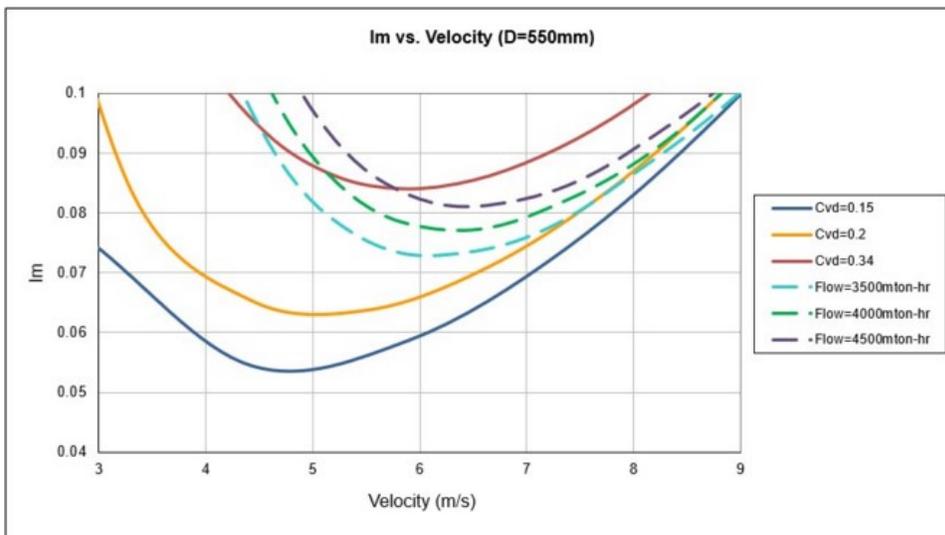
- FthVerify75 - 550 mm.fth
- FthVerify75 - 650 mm.fth
- FthVerify75 - 750 mm.fth

REFERENCE: Wilson, Addie, Sellgren & Clift, Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer, Pages 329-337, Case Study 13.1

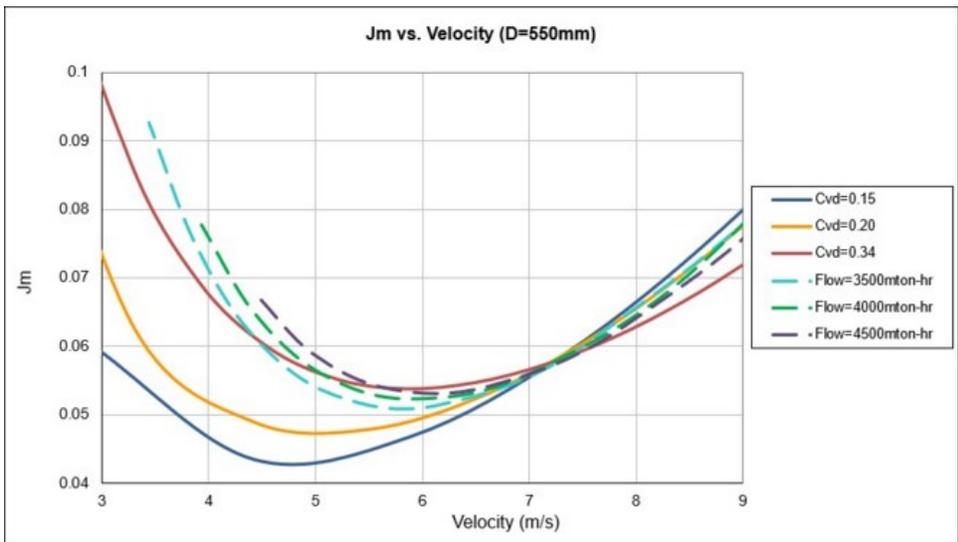
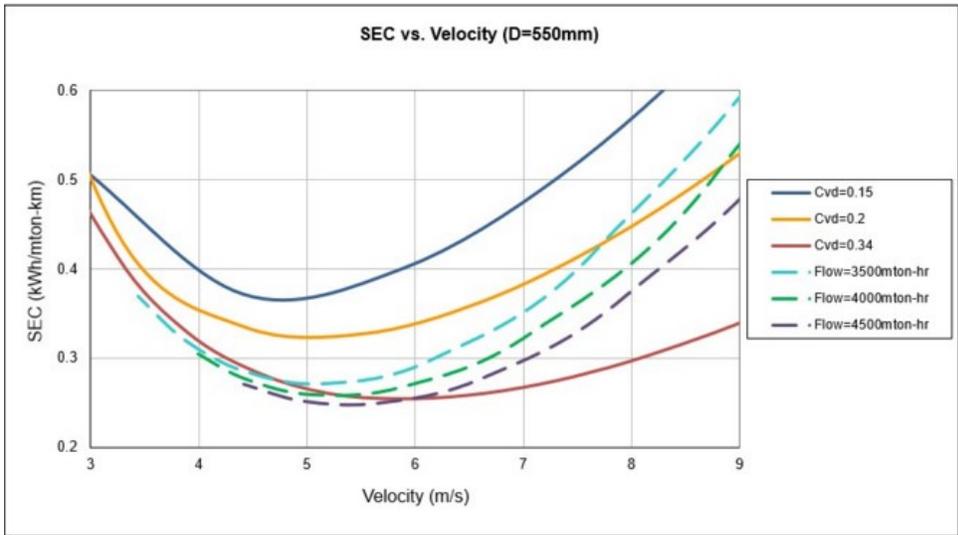
FLUID: Water/Sand

ASSUMPTIONS: Water temperature is assumed to be 4 deg. C. Other than the pipe diameter, the system configuration is not given (i.e. pressures, pipe length, etc) and were assumed. These assumptions do not affect the final results of the problem. Assume $V_t/V_{ts} = 0.56$ for sand.

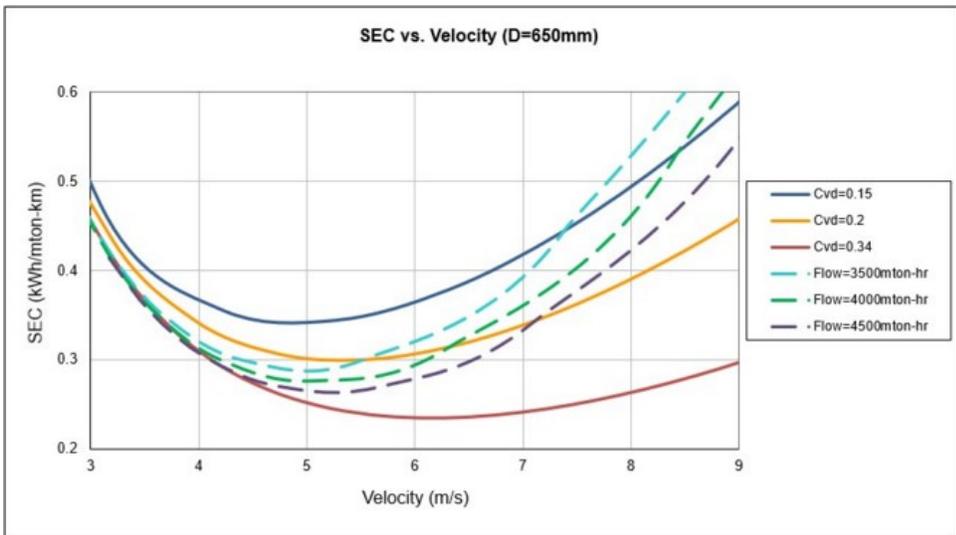
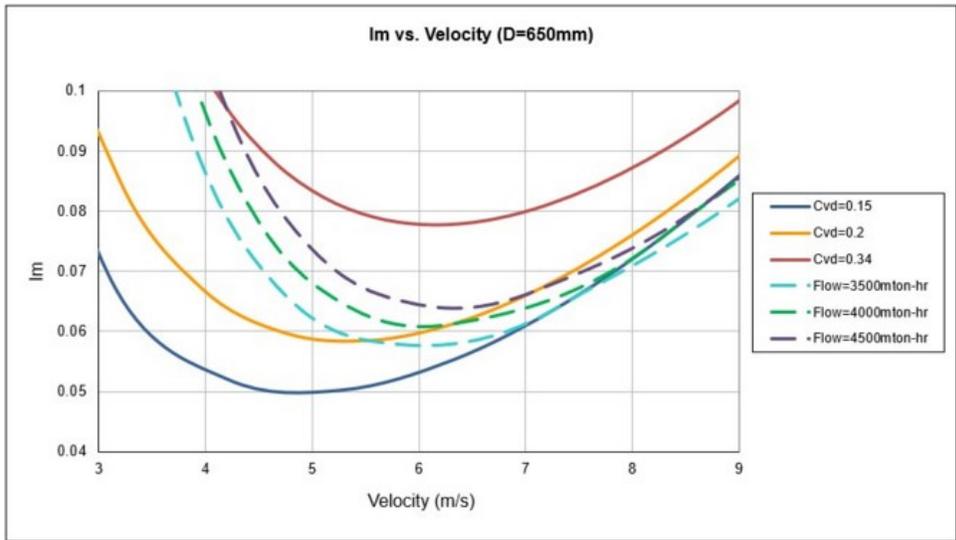
RESULTS:



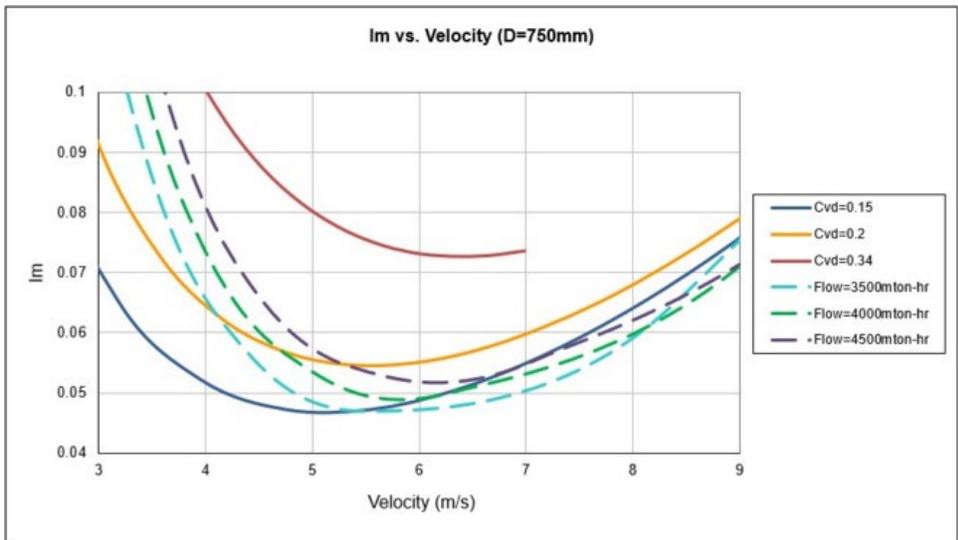
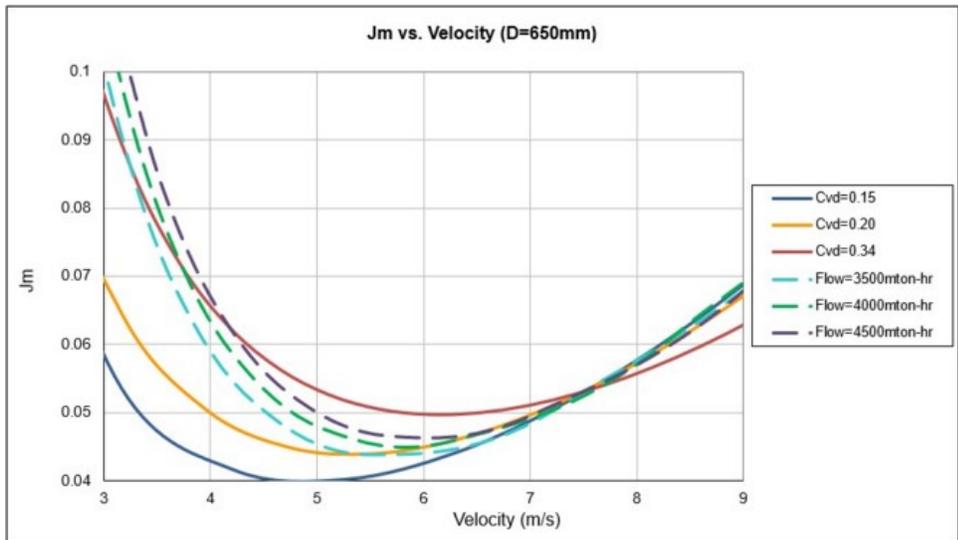
Verification Case 75



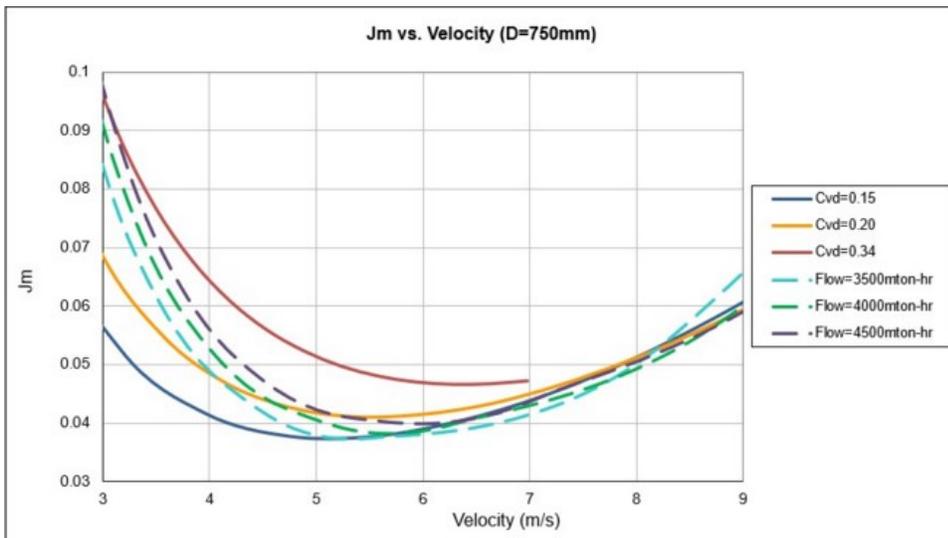
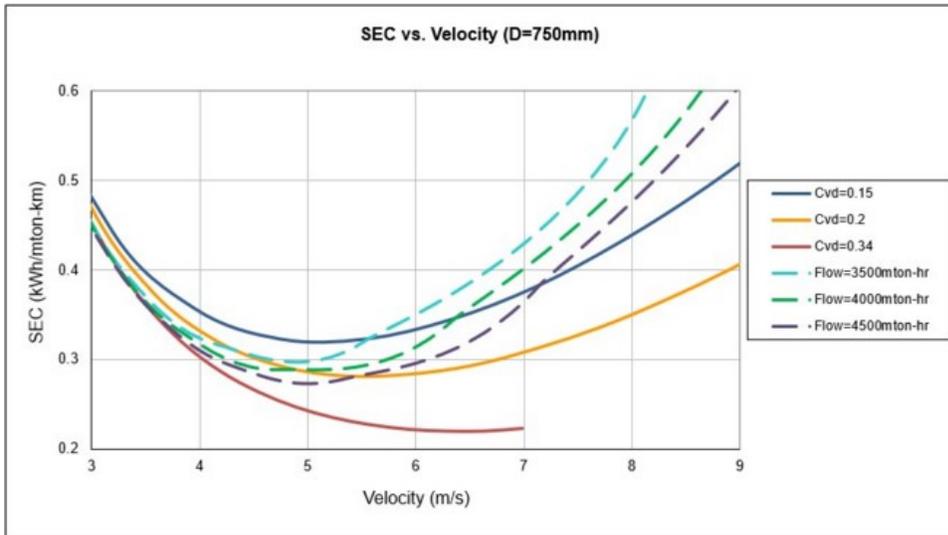
Verification Case 75



Verification Case 75



Verification Case 75



[List of All Verification Models](#)

Verification Case 75 Problem Statement

[Verification Case 75](#)

Wilson, Addie, Sellgren & Clift, Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer, Pages 329-337, Case Study 13.1

[Wilson, Addie, Sellgren & Clift's Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

13. System Design and Operability

329

13.5 Case Study

Case Study 13.1

Case Studies 5.1 and 6.1 were concerned with the transport of 4000 tonnes per hour of coarse sand ($S_s = 2.65$) as a slurry in water, at delivered concentrations up to $C_{vd} = 0.24$. In this case study the design will be completed by examining system operability. Because the earlier studies all represent stages in the design, they will be summarised here.

Case Study 5.1 was the preliminary selection of pipe size, not needing information on the particle size beyond noting that the slurry has settling characteristics. Three pipe sizes were identified as suitable for more detailed study, keeping $V_m > V_{sm}$ to avoid deposition. The results are shown on Table 13.1.

Table 13.1

Table 13.1 Condition	D (m)	V_{sm} (m/s)	C_{vd}	V_m (m/s)
A	0.55	4.45	0.24	7.35
B	0.60	4.65	0.24	6.17
C	0.65	4.84	0.20	6.30

In Case Study 6.1 the particle size distribution was introduced, and used to calculate the friction gradient and specific energy consumption. It was shown that, for the 0.65 m pipe with 20% solids of the type specified (Condition C above) the friction gradient is 0.0612 m. water/m. pipe with specific energy consumption (SEC) of 0.315 kWh/tonne-km. It was also found that increasing C_{vd} and decreasing V_m reduced the specific energy consumption. A revised operating point (Condition D) was therefore chosen for further analysis.

D = 0.65 m ($V_{sm} = 4.84$ m/s)
C_{vd} = 0.221
V_m = 5.70 m/s
i_m = 0.0619 m.water/m.pipe
SEC = 0.289 kWh/tonne-km

The feasibility of operating at this point was not investigated in Case Study 6.1. Note that for these conditions the 'standard velocity' V_x is 7.52 m/s

(from Eq. 13.1) and the velocity at minimum point V_o is 5.48 m/s (from Eq. 13.3). Thus the proposed operating velocity V_m lies between V_o and V_x . It is presumed that the pumps have been selected to operate at V_m and that they have the usual falling characteristics: i.e. head developed decreases with discharge.

We now pursue this study further by examining system operability. Figures 13.9 to 13.14 show the system characteristics and SEC curves for three pipe sizes: 0.65, 0.75 and 0.55 m.

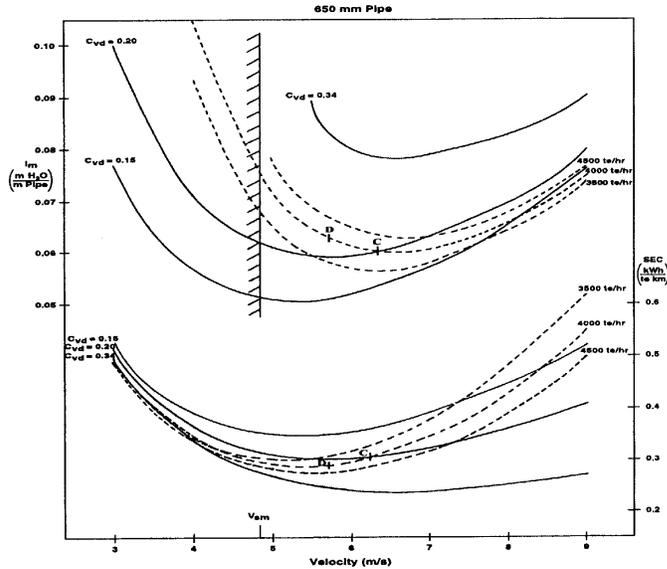


Figure 13.9. Friction gradient (i_m) and specific energy consumption (SEC) for transport of coarse sand in 0.65 m pipe

The curves are all calculated using the approach illustrated in Case Study 6.1, with values for the deposit velocity (V_{sm}) obtained from Eq. 5.10 as in Case Study 5.1. We consider first the pipe size which appeared to be attractive in the earlier case studies : $D = 0.65$ m. Figure 13.9 shows the system characteristics (expressed in terms of i_m) and SEC curves. The full curves are drawn for constant delivered concentration. The broken curves

refer to constant throughput: throughput can be kept constant by increasing mixture velocity while decreasing delivered concentration. Conditions C and D considered in earlier case studies correspond to the points indicated on this figure.

Only now that the system characteristics are plotted is it evident that Condition D is too far back on the system curves to represent a feasible operating point for this material - the system characteristic shows i_m decreasing as V_m increases, so that operation will be unstable. Even Condition C is marginal. Given that the booster pump will have variable-speed diesel drive, operation at C might be feasible if solids consistency and throughput do not fluctuate too widely. However, if the variations are significant - for example, throughput variations from 3500 tonne/hr to 4500 tonne/hr (see Figure 13.9) - then operation will become unstable.

To achieve truly flexible operation, it is preferable to operate around the 'standard' velocity, where $i_w = j_m$. Figure 13.10 shows the same system characteristics, but now plotted in terms of j_m (i.e. i_m/S_m in m. slurry/m. pipe). The recommended operating velocity is therefore around point E, i.e. at V_m of 7.54 m/s, with $C_{vd} = 0.168$ and with SEC increased to 0.421 kWh/tonne-km. Thus, once operability is considered, it is seen that lower concentrations should be used, with correspondingly higher energy consumption.

We consider now whether going to a larger pipe would improve matters. Figures 13.11 and 13.12 show the curves of i_m , SEC and j_m for a pipe of 0.75 m diameter. To achieve operation at the standard velocity, it is now necessary to reduce the concentration further still: $V_m = 7.83$ m/s, with $C_{vd} = 0.121$. Again because of the cost of pumping the conveying water, the specific energy consumption is high: SEC is 0.511 kWh/tonne-km at a throughput of 4000 tonnes per hour. Furthermore, increasing the pipe size will increase the capital cost. Thus the 0.75 m pipe is not recommended.

A smaller pipe size should also be considered. Figures 13.13 and 13.14 show the curves of i_m , SEC and j_m calculated in the usual way for a pipe 0.55 m in diameter. Condition A from Case Study 5.1 is now seen to be quite close to the standard velocity. It therefore represents a flexible but stable operating point. Furthermore, because the delivered concentration is relatively high, at about 24% solids, the specific energy consumption is quite attractive, at 0.353 kWh/tonne-km.

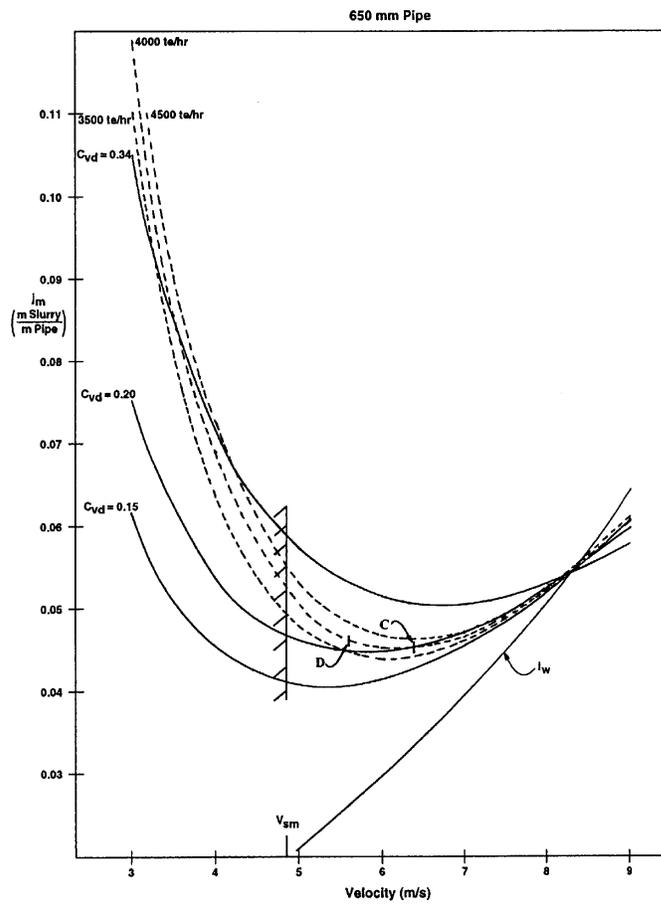


Figure 13.10. Friction gradient in terms of head of slurry (j_m) for coarse sand in 0.65 m pipe

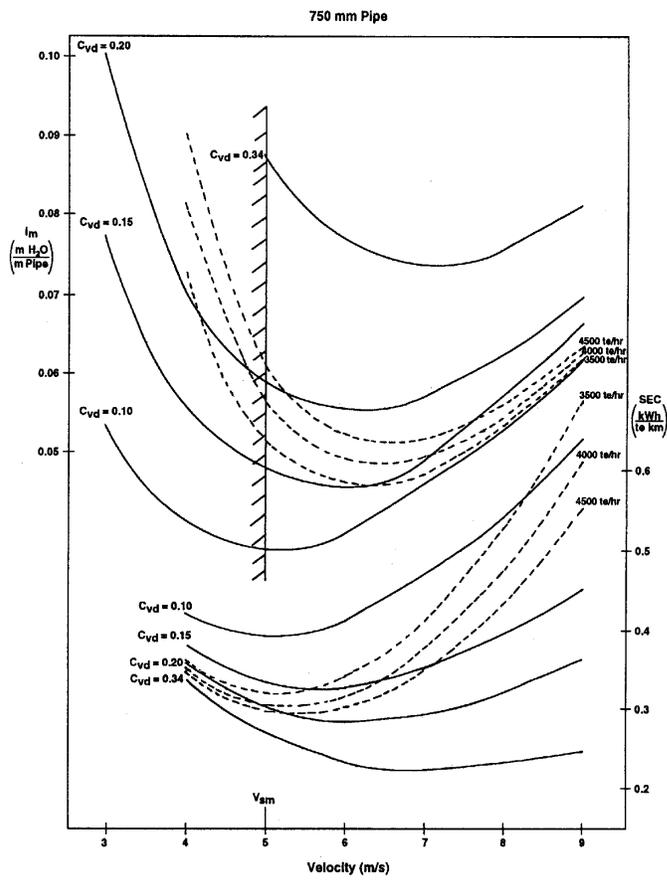


Figure 13.11. Friction gradient (i_m) and specific energy consumption (SEC) for transport of coarse sand in 0.75 m pipe

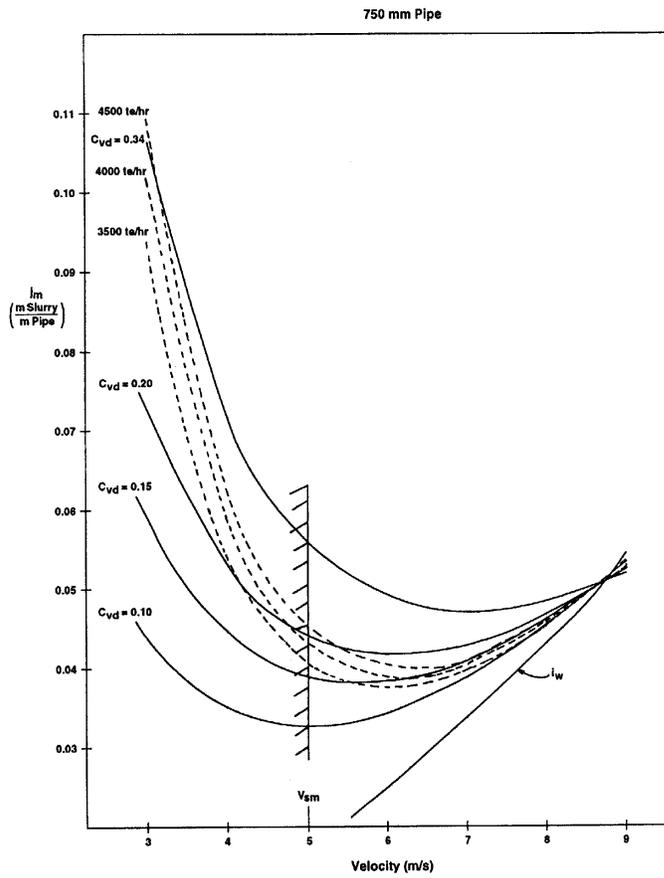


Figure 13.12. Friction gradient in terms of head of slurry (j_m) for coarse sand in 0.75 m pipe

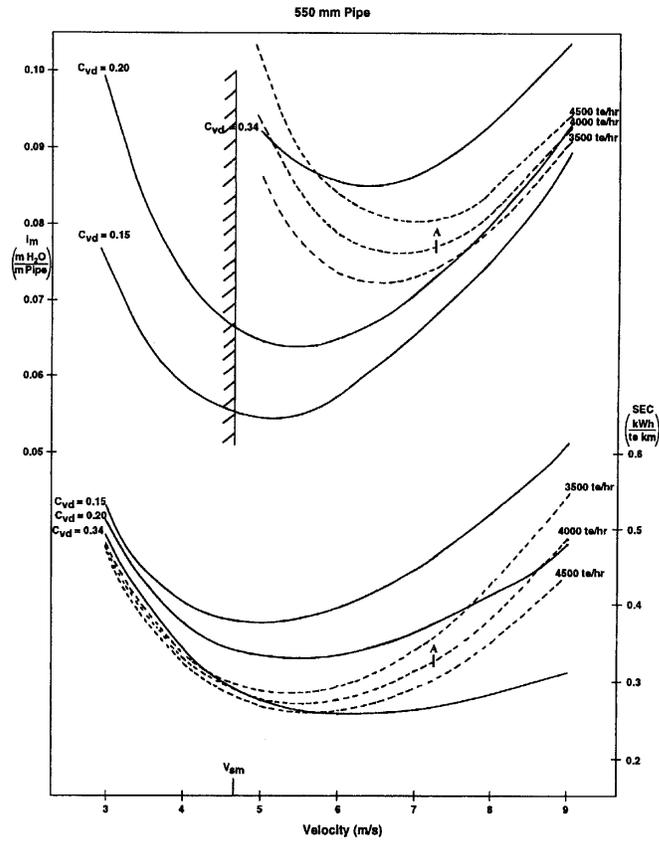


Figure 13.13. Friction gradient (i_m) and specific energy consumption (SEC) for transport of coarse sand in 0.55 m pipe

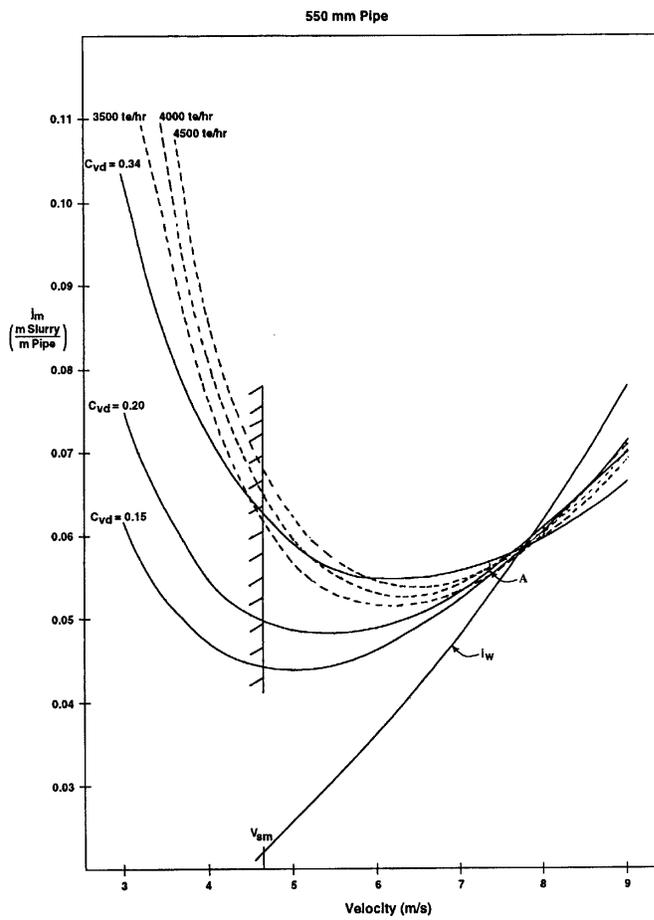


Figure 13.14. Friction gradient in terms of head of slurry (j_m) for coarse sand in 0.55 m pipe

Thus, of the three pipe sizes considered in this case study, the smallest would be preferred. In addition to its favorable specific energy consumption, it will have the least capital cost. It is left to the reader to carry out the comparison for the intermediate size, $D = 0.60$ m. A full design would now proceed to consider pump selection again, essentially repeating Case Study 12.1 for the revised pipe size.

The conclusions illustrated by this case study are general: for slurries with settling characteristics, considerations of energy consumption together with operational stability favour use of high solids concentration and small pipe diameter. Also, as indicated throughout this chapter, stable operation rather than deposition controls the operating velocity for this type of slurry.

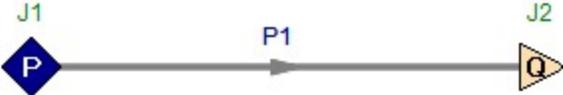
On the negative side, the mixture velocity can be too high. For example, Condition A, as selected above, has a design mixture velocity of 7.35 m/s, and the pipe will experience substantial wear. As noted in Chapter 11, for some applications, e.g. conveying of phosphate matrix, wear is accommodated by rotating the pipe at intervals, usually through 120° , so as to expose new parts of the pipe wall to the stratified solids which cause the abrasion.

REFERENCES

- Spells, K.E. (1955). Correlations for use in transport of aqueous suspensions of fine solids through pipes. *Trans. Instn. Chem. Engrs.*, Vol. 33, pp. 79-84.
- Wilson, K.C., Clift, R. & Sellgren, A. (2002). Operating points for pipelines carrying concentrated heterogeneous slurries. *Powder Technology*, Vol. 123, pp. 19-24.

View Verification Case 75 Model

[Verification Case 75](#)



Verification Case 76

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom (with SSL Module)

TITLE: FthVerify765.fth

REFERENCE: Wilson, Addie, Sellgren & Clift, Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer, Pages 117-118, Case Study 5.1

FLUID: Water/Sand

ASSUMPTIONS: Water temperature is assumed to be 20 deg. C. Other than the pipe diameter, the system configuration is not given (i.e. pressures, pipe length, etc) and were assumed. These assumptions do not affect the final results of the problem. Use the Minimal slurry calculation method. Assume the sliding friction coefficient is 0.4.

RESULTS:

Wilson, Addie, Sellgren & Clift Results:

Case	Velocity (meters/sec)	Settling Velocity (meters/sec)
D = 0.60 m	6.17	4.65
D = 0.55 m	7.35	4.45
D = 0.65 m	5.26	4.84

AFT Fathom Results:

Case	Velocity (meters/sec)	Settling Velocity (meters/sec)
D = 0.60 m	6.18	4.65
D = 0.55 m	7.36	4.45
D = 0.65 m	5.27	4.84

DISCUSSION:

The results from AFT Fathom match the results from Wilson, Addie, Sellgren & Clift very closely. It should be noted that the settling velocities for the 0.55 m and 0.65 m pipe diameter cases are assumed to be misprinted in the reference as 5.45 m/s and 5.84 m/s, because these cases should "bound" the 0.60 m case.

[List of All Verification Models](#)

Verification Case 76 Problem Statement

[Verification Case 76](#)

Wilson, Addie, Sellgren & Clift, Slurry Transport Using Centrifugal Pumps 3rd Edition, 2006, Publisher Springer, Pages 329-337, Case Study 13.1

[Wilson, Addie, Sellgren & Clift's Title Page](#)

Note: If you are having trouble reading the scanned image below, it is usually much easier to read when printed.

5. Motion and Deposition of Settling Solids

117

covered in the following chapter, will generally display smaller values of the solids effect.

Another type of stratified flow that may be encountered is flow above a stationary bed of solids. This type of operation is generally uneconomic, and thus is seldom a deliberate design choice, but it is sometimes found in existing pipelines. Flow over a stationary bed has been studied experimentally, and analysed by a computer model (Nnadi & Wilson, 1992; Pugh, 1995). For most cases of this type of flow encountered in pipelines a rough approximation to the computer output may be sufficient. In such instances, the following equation for estimating i_m may be useful. It is based on particles near the 'Murphian' size, which tend to be disproportionately represented in stationary deposits

$$i_m = 0.32 (S_s - 1)^{1.05} C_{vd}^{0.6} \left(\frac{V_m}{(2gD)^{1/2}} \right)^{0.1} \quad (5.14)$$

In some cases it will be necessary to incorporate the effects of pipe slope, which will be dealt with in Chapter 8. Before considering sloping flows, however, it will be necessary to investigate fluid suspension of particles, and show how suspended load combines with contact load in heterogeneous slurry flow. These are the subjects of Chapter 6.

5.6 Case Studies

Case Study 5.1 - Preliminary Pipe Sizing

A pipe is to be designed to convey solids at 4000 tonnes/hour (on a dry-weight basis). The material is described only as 'coarse sand', and it is estimated that the largest volumetric solids concentration which can be fed to the pumps is $C_{vd} = 0.24$. Preliminary estimates are required of suitable combinations of pipe size, C_{vd} and V_m . For convenience it will be assumed that pipes are available with inside diameter D in increments of 50 mm.

First, the tonnes per hour is converted to m^3/s of solids, Q_s , by dividing by the estimated sand density of 2.65 tonnes/ m^3 and by 3600 s/hour, giving $Q_s = 0.419 m^3/s$. The mixture discharge Q_m equals Q_s/C_{vd} and hence the minimum Q_m , corresponding to the maximum C_{vd} of 0.24, equals 1.747 m^3/s .

The next step is to select a pipe size which gives a reasonable velocity for this value of Q_m . For example, a pipe of $D = 0.60$ m has a cross-sectional area of 0.283 m^2 , and hence the velocity in this pipe must be at least 1.747/0.283 or 6.17 m/s in order to satisfy the maximum - C_{vd} condition.

This velocity seems to be in the right 'ball park', but must be compared with the deposition velocity V_{sm} . The particle size distribution of the coarse sand may not be known, but it is safe to assume that it contains grains of the 'Murphian' size, which is about 0.7 mm for this pipe diameter. For this particle and pipe size combination, the nomographic chart (Fig. 5.3) gives $V_{sm} \approx 5.8$ m/s, but it should be recalled that the value of V_{sm} estimated from the chart tends to be high. This point can be checked by means of Eq. 5.11, using a reasonable value of f_w , say 0.012. The result is a revised value of V_{sm} of only 4.65 m/s, so that the minimum operating velocity of 6.17 m/s provides a generous margin against deposition.

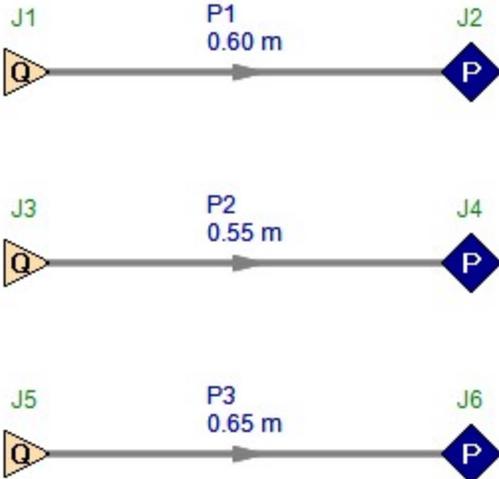
Other pipe sizes should also be investigated. The next smallest pipe ($D = 0.55$ m), is found to have a minimum operating velocity of 7.35 m/s, based on $C_{vd} = 0.24$. This can be compared to V_{sm} values of 5.4 m/s from Fig. 5.3 and 5.45 m/s from Eq. 5.11. Using the latter number as the upper limit for deposition, it is seen that the minimum operating velocity would be 65 percent in excess of the deposition limit, producing stable operation at the cost of high frictional losses. This pipe size could merit further investigation of the sort illustrated in case studies in subsequent chapters, but it is obvious that no smaller pipe would be suitable, and the 0.55 m pipe appears at present to be considerably less attractive than the 0.60 m size.

Turning to a larger pipe, with $D = 0.65$ m, it is found that for $C_{vd} = 0.24$ the minimum velocity is 5.26 m/s, while the V_{sm} values are 6.1 m/s from Fig. 5.3 and 5.84 m/s from Eq. 5.11. The minimum operating velocity of 5.26 m/s is less than 10 percent in excess of V_{sm} by Eq. 5.11, which is not a sufficient margin. A margin of 30 percent should be adequate, giving $V_m \approx 6.30$ m/s. Increasing the velocity to this value requires a lower solids concentration, i.e. 0.20 instead of 0.24. This combination ($D = 0.65$ m, $V_m = 6.3$ m/s, $C_{vd} = 0.20$) merits further investigation, but any larger pipe size (which would involve higher operating velocity and lower delivered concentration), would probably not be suitable.

It is worth noting that this case study shows that a preliminary selection of appropriate pipe sizes can be made on the basis of criteria of continuity and deposition velocity, even though very little is known about the solids in the slurry. In the present instance, it is found that to transport 4000 tonnes of coarse sand per hour, only pipes of internal diameter 0.60 m and 0.65 m appear to be attractive.

View Verification Case 76 Model

[Verification Case 76](#)



Verification Case 77

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom (with ANS Module)

TITLE: FthVerify77.fth

REFERENCE: William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Example 5.8, pages 179-180.

FLUID: Methanol

ASSUMPTIONS: N/A

RESULTS:

	Janna	AFT Fathom
Minimum Diameter	6.338 cm	6.337 cm
Nominal Size	2-1/2 in	2-1/2 in

DISCUSSION:

The AFT Fathom model used an assigned pressure upstream and a receiving reservoir at atmospheric pressure. The only design requirement for the system was a minimum flow rate of 0.8 m³/min.

The automated sizing was a discrete sizing to minimize flow volume.

[List of All Verification Models](#)

Verification Case 77 Problem Statement

Verification Case 77

William S. Janna, Introduction to Fluid Mechanics, 1983, PWS Publishers, Example 5.8, pages 179-180.

Janna's Title Page

Because V_2 and f are unknown, a trial and error solution is required. As a first trial assume $f = 0.03$; then

$$\left. \begin{aligned} V_2 &= 6.72 \text{ ft/s} \\ \text{Re} &= \frac{\rho V D}{\mu g_c} = \frac{62.4(6.72)(0.1342)}{6 \times 10^{-4}} = 9.38 \times 10^4 \\ \frac{\epsilon}{D} &= \frac{0.00085}{0.1342} = 0.0063 \end{aligned} \right\} \begin{array}{l} \text{Moody diagram:} \\ f = 0.0335 \end{array}$$

As a second trial assume $f = 0.0335$; then

$$\left. \begin{aligned} V_2 &= 6.51 \text{ ft/s} \\ \text{Re} &= 9.1 \times 10^4 \\ \frac{\epsilon}{D} &= 0.0063 \end{aligned} \right\} \begin{array}{l} \text{Moody diagram} \\ f = 0.034 \quad (\text{close enough}) \end{array}$$

Using $f = 0.034$, $V_2 = 6.49$ ft/s, and the continuity equation, we find

$$Q = A_2 V_2 = 0.01414(6.49)$$

$$Q = 0.0918 \text{ ft}^3/\text{s} = 5.51 \text{ ft}^3/\text{min}$$

EXAMPLE 5.8

Methyl alcohol is used in a processing plant where a flow rate of $0.8 \text{ m}^3/\text{min}$ of the liquid must be supplied. The available liquid pump can supply this flow rate only if the pressure drop in the supply line is less than 15 m head of water. The pipeline is made up of 50 m of soldered drawn copper tubing and follows the path shown in Figure 5.24. Determine the minimum size of tubing required.

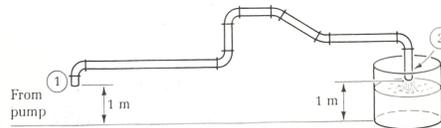


Figure 5.24. Sketch for Example 5.8.

SOLUTION

Write the Bernoulli equation from 1 to 2 as

$$\frac{p_1 g_c}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{p_2 g_c}{\rho g} + \frac{V_2^2}{2g} + z_2 + \sum \frac{fL}{D} \frac{V^2}{2g} + \sum K \frac{V^2}{2g}$$

Verification Case 77 Problem Statement

The pressure drop is given in terms of meters of water:

$$\frac{p_1 - p_2}{\rho} \frac{g_c}{g} = 15 \text{ m of water}$$

where ρ is for water. From Appendix Table A-5 for methyl alcohol, $\rho = 789 \text{ kg/m}^3$ and $\mu = 0.56 \times 10^{-3} \text{ N-s/m}^2$. To convert the pressure drop to meters of alcohol, multiply by the ratio of densities:

$$\frac{p_1 - p_2}{\rho} \frac{g_c}{g} = \frac{15(1000)}{789} = 19.01 \text{ m of methyl alcohol}$$

The velocity at point 1 is the same as that at point 2 because the area at these points is the same. Moreover, $z_1 = z_2$. Thus the Bernoulli equation becomes

$$19.01 = \frac{V^2}{2g} \left(\frac{fL}{D} + \Sigma K \right)$$

From continuity, we have $V = 4Q/\pi D^2$. From Table 5.4, we find the minor losses for four 90° elbows (assumed regular because nothing special was specified) and two 45° elbows; for lack of more specific data, take the minor loss coefficients to be the same as those for threaded fittings. Note that an exit fitting is not included because the exit kinetic energy is not zero. Therefore,

$$\Sigma K = 4(1.4) + 2(0.35) + 1.0 = 6.3$$

From Table 5.2 for drawn tubing, $\epsilon = 0.00015 \text{ cm}$. By substitution into the Bernoulli equation, we get

$$19.01 = \frac{16Q^2}{2g\pi^2 D^5} \left[\frac{f(50)}{D} + 6.3 \right]$$

With $Q = 0.8 \text{ m}^3/\text{min} = 0.013 \text{ m}^3/\text{s}$, the equation becomes

$$\frac{19.01(2)(9.81)\pi^2}{16(0.013)^2} = \frac{f(50)}{D^5} + \frac{6.3}{D^4}$$

or

$$3.67 \times 10^{-5} f = D^5 - 4.63 \times 10^{-6} D$$

A trial and error approach is required, but in this case it is easier to assume a diameter. As a first trial assume $D = 0.05 \text{ m}$; then

$$f = 0.0012 \quad (\text{by equation above; } 0.0012 \text{ is unlikely—not on Moody diagram})$$

As a second trial assume $D = 0.06 \text{ m}$; then

$$f = 0.014 \quad (\text{by equation})$$

$$A = \frac{\pi D^2}{4} = 0.00283 \text{ m}^2$$

$$\begin{aligned}
 V &= \frac{Q}{A} = \frac{0.013}{0.00283} = 4.60 \text{ m/s} \\
 \text{Re} &= \frac{\rho V D}{\mu g_c} = \frac{789(4.6)(0.06)}{0.56 \times 10^{-3}} \\
 &= 3.89 \times 10^5 \\
 \frac{\epsilon}{D} &= \frac{0.00015 \text{ cm}}{6.0 \text{ cm}} = 0.000025
 \end{aligned}
 \left. \vphantom{\begin{aligned} V \\ \text{Re} \\ \frac{\epsilon}{D} \end{aligned}} \right\} \begin{array}{l} \text{Moody diagram:} \\ f = 0.014 \end{array}$$

The f value calculated by equation is slightly less than f from the Moody diagram. At this point, then, rather than using randomly chosen values of D , we use Appendix Table C-2 values for type M copper tubing (soldered fittings). For our next trial we therefore assume that $D = 6.338$ cm (slightly larger than the 6-cm diameter of the second trial), corresponding to the $2\frac{1}{2}$ -standard type M. As a third trial assume $D = 0.06338$ m; then

$$\begin{aligned}
 A &= 0.004017 \text{ m}^2 \quad (\text{from Appendix Table C-2}) \\
 f &= 0.020 \quad (\text{by equation}) \\
 V &= 3.236 \text{ m/s} \\
 \text{Re} &= 2.74 \times 10^5 \\
 \frac{\epsilon}{D} &= 0.000024
 \end{aligned}
 \left. \vphantom{\begin{aligned} A \\ f \\ V \\ \text{Re} \\ \frac{\epsilon}{D} \end{aligned}} \right\} \begin{array}{l} \text{Moody diagram:} \\ f = 0.015 \end{array}$$

Thus the calculated value of $f = 0.020$ is greater than that from the Moody diagram ($f = 0.015$) for the $2\frac{1}{2}$ -standard type M copper tube. Anything smaller will reduce the volume flow rate; anything larger will work but incurs an unnecessary expense. Thus select

$$\underline{D = 6.338 \text{ cm} \quad (2\frac{1}{2}\text{-standard type M copper tube)}$$

5.8 / Hydraulic and Energy Grade Lines

Hydraulic and energy grade lines give a graphic presentation of the flow quantities in a particular configuration. The **hydraulic grade line** is a plot of pressure versus distance; the **energy grade line** is a plot of the sum of pressure and kinetic energy as a function of distance. These concepts are best illustrated by example.

Consider two large reservoirs connected with a pipe as shown in Figure 5.25. Because one liquid reservoir surface is elevated above the other, a flow exists in the pipe. The total mechanical energy in the system at section 1 is

$$E_1 = \frac{\rho_1 g_c}{\rho g} + \frac{V_1^2}{2g} + z_1$$

View Verification Case 77 Model

[Verification Case 77](#)



Verification Case 78

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom (with ANS Module)

TITLE: FthVerify78.fth

REFERENCE: James John, William Haberman, Introduction to Fluid Mechanics, 2nd Ed., 1980, Prentice-Hall, Example 6.6, page 179.

FLUID: Water

ASSUMPTIONS: N/A

RESULTS:

	John & Haberman	AFT Fathom Continuous	AFT Fathom Discrete
Minimum Diameter	0.0311 m	0.02848 m	0.03505 m
Nominal Size	-	-	1-1/4 in

DISCUSSION:

The AFT Fathom pipe material library only has ductile iron data from 3 inch to 54 inch. The minimized pipe size is less than 3 inch so steel pipe was substituted for cast iron. Because the roughness of steel pipe is less than cast iron, the minimized size for the continuous case was smaller than the reference value.

Both the continuous and discrete automated sizing scenarios were run to minimize flow volume.

[List of All Verification Models](#)

Verification Case 78 Problem Statement

Verification Case 78

James John, William Haberman, Introduction to Fluid Mechanics, 2nd Ed., 1980, Prentice-Hall, Example 6.6, page 179.

John & Haberman's Title Page

Sec. 6.5

Piping Systems

179

determined; a trial-and-error solution is called for. As a first trial, assume $f = 0.02$.

$$(9.81 \text{ m/s}^2)(20 \text{ m}) = \left[\frac{0.02 \times 250}{0.10} + \left(\frac{0.5}{\text{square-edged inlet}} + 3.0 + \frac{6.4}{\text{globe valve}} \right) + 1 \right] \frac{V_c^2}{2}$$

$$\frac{V_c^2}{2} = \frac{9.81(20)}{50 + 9.9 + 1} \text{ m}^2/\text{s}^2$$

or

$$V_c = 1.795 \text{ m/s}$$

For this velocity,

$$\text{Re} = \frac{VD}{\nu} = \frac{(1.795 \text{ m/s})(0.10 \text{ m})}{1.00 \times 10^{-6} \text{ m}^2/\text{s}} = 1.795 \times 10^5$$

$$\frac{\epsilon}{D} = \frac{0.12 \times 10^{-3} \text{ m}}{0.10 \text{ m}} = 1.2 \times 10^{-3}$$

From the Moody diagram, we obtain $f = 0.022$. Since this value does not agree with our initial trial, we shall now make a second trial, starting with $f = 0.022$.

$$\frac{fL}{D} = \frac{0.022 \times 250}{0.10} = 55$$

$$\frac{V_c^2}{2} = \frac{9.81(20)}{55 + 9.9 + 1}$$

or

$$V_c = 1.725 \text{ m/s}$$

For this velocity,

$$\text{Re} = \frac{VD}{\nu} = \frac{1.725 \times 0.10}{10^{-6}} = 1.725 \times 10^5$$

From the Moody diagram, we find $f = 0.022$. It can be seen that generally we are able to converge on the correct answer quite rapidly; no more than two trials are required, as a rule. The water flow rate is

$$Q = AV$$

$$= \left[\frac{\pi}{4} (0.10)^2 \text{ m}^2 \right] 1.725 \text{ m/s}$$

$$= 0.01355 \text{ m}^3/\text{s}$$

$$= 13.55 \text{ l/s}$$

EXAMPLE 6.6. A pump is to be used to supply 5 liters per second of water from a reservoir to a point 400 m from the reservoir at the same level as the reservoir surface. Determine the minimum-size cast iron pipe required. The water temperature is 15°C; assume that minor losses can be neglected. (See Figure 6.33.) The pump supplies 50 kW of power to the water flow.

Solution: From (6.8) we can express the pump work in terms of reservoir surface and outlet conditions:

$$-\frac{1}{\dot{m}} \frac{dW_p}{dt} = \frac{p_e - p_0}{\rho} + g(z_e - z_0) + \frac{fL}{D} \frac{V^2}{2} + \frac{V_e^2}{2}$$

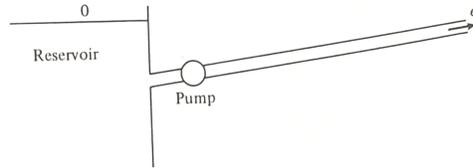


Figure 6.33

For our case,

$$\dot{m} = \rho Q = (999.2 \text{ kg/m}^3)(5 \times 10^{-3} \text{ m}^3/\text{s}) = 4.996 \text{ kg/s}$$

$$-\frac{1}{\dot{m}} \frac{dW_p}{dt} = -\frac{1}{4.996 \text{ kg/s}} (-50 \times 10^3 \text{ N}\cdot\text{m/s}) = +10,010 \text{ m}^2/\text{s}^2$$

(Note: dW_p/dt is negative work in a thermodynamic sense, for it represents work done on the fluid.)

We now have

$$10,010 = \left(\frac{fL}{D} + 1 \right) \frac{V^2}{2}$$

where

$$\begin{aligned} V &= \frac{Q}{A} = \frac{5 \times 10^{-3} \text{ m}^3/\text{s}}{(\pi/4)D^2 \text{ m}^2} \\ &= \frac{6.366 \times 10^{-3}}{D^2} \text{ m/s} \quad \text{with } D \text{ in m} \\ 10,010 &= \left(\frac{400f}{D} + 1 \right) \frac{4.053 \times 10^{-5}}{2D^4} \end{aligned}$$

For this case, with D the unknown, the Reynolds number and f cannot be immediately determined. A trial-and-error procedure is required. As a first trial, assume that $f = 0.025$.

$$\begin{aligned} 10,010 &= \left[\frac{400(0.025)}{D} + 1 \right] \frac{2.026 \times 10^{-5}}{D^4} \\ &= \frac{20.26 \times 10^{-5}}{D^5} + \frac{2.026 \times 10^{-5}}{D^4} \end{aligned}$$

The second term on the right is small compared to the first, so, to a first approximation,

$$10,010 \sim \frac{2.026 \times 10^{-4}}{D^5} \quad \text{and} \quad D = 0.0289 \text{ m}$$

For this first trial,

$$V = \frac{6.366 \times 10^{-3}}{(0.0289)^2} = 7.622 \text{ m/s}$$

$$\text{Re} = \frac{7.622 \times 0.0289}{1.15 \times 10^{-6}} = 1.915 \times 10^5$$

$$\frac{\epsilon}{D} = \frac{0.26 \times 10^{-3}}{0.0289} = 0.0090$$

From the Moody diagram, we obtain $f = 0.036$. To start a second iteration, let $f = 0.036$. Therefore,

$$10,010 = \left[\frac{400(0.036)}{D} + 1 \right] \frac{2.026 \times 10^{-5}}{D^4}$$

or

$$10,010 \sim \frac{0.0002917}{D^5} \quad \text{and} \quad D = 0.0311 \text{ m}$$

For this second trial,

$$V = \frac{6.366 \times 10^{-3}}{(0.0311)^2} = 6.582 \text{ m/s}$$

$$\text{Re} = \frac{6.582 \times 0.0311}{1.15 \times 10^{-6}} = 1.78 \times 10^5$$

$$\frac{\epsilon}{D} = \frac{0.26 \times 10^{-3}}{0.0311} = 0.0084$$

From the Moody diagram, we obtain $f = 0.036$. The agreement with the assumed value is good enough so that $D = 0.0311 \text{ m}$ can be taken as the required diameter.

6.6 PIPES IN PARALLEL

The previous section has dealt with systems in which the resistances (friction and minor losses) were placed in series. In many cases, however, a system involves pipes in parallel, as shown in Figure 6.34. For example, it might be

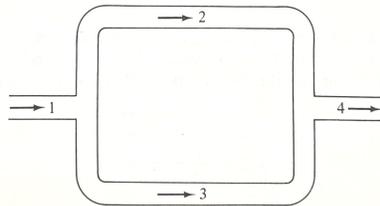


Figure 6.34

required to find the flows through 2 and 3 for given pipe diameters D_1 , D_2 , D_3 , and D_4 , given $p_1 - p_4$, and given flow at 1. In this case, the flow resistances are in parallel. There is a direct analog to a dc electrical circuit. In an electrical circuit, electrical current (amperes) flows through an electrical resistance measured in ohms. The driving force for such a system is voltage drop. In the circuit shown (Figure 6.35) resistances R_2 and R_3 are in parallel.

View Verification Case 78 Model

[Verification Case 78](#)



Verification Case 79

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom (with ANS Module)

TITLE: FthVerify79.fth

REFERENCE: Robert W. Fox and Alan T. McDonald, Introduction to Fluid Mechanics, 3rd Ed., 1985, John Wiley & Sons, Example 8.8, page 377.

FLUID: Water

ASSUMPTIONS: N/A

RESULTS:

	Fox & McDonald	AFT Fathom
Minimum Diameter	6.065 in	6.065 in
Nominal Size	6 inch	6 inch

DISCUSSION:

The AFT Fathom pipe material library does not have drawn aluminum pipe so steel pipe size data was used. The roughness model was modified to use a relative roughness of 0.00001, which is representative of drawn tubing. With this setting, the results for nominal size matched exactly.

The automated sizing was discrete with the objective set to minimize flow volume.

[List of All Verification Models](#)

Verification Case 79 Problem Statement

Verification Case 79

Robert W. Fox and Alan T. McDonald, Introduction to Fluid Mechanics, 3rd Ed., 1985, John Wiley & Sons, Example 8.8, page 377.

Fox & McDonald's Title Page

quite old, choose $e/D = 0.005$. Then, from Fig. 8.14, guess $f \approx 0.03$. Then a first approximation to \bar{V}_2 is

$$\bar{V}_2 = \left[2 \times \frac{32.2 \text{ ft}}{\text{sec}^2} \times 80 \text{ ft} \times \frac{1}{0.03(2040 + 8) + 1} \right]^{1/2} = 9.08 \text{ ft/sec}$$

Now check the value assumed for f .

$$Re = \frac{\rho \bar{V} D}{\mu} = \frac{\bar{V} D}{\nu} = \frac{9.08 \text{ ft}}{\text{sec}} \times \frac{\text{ft}}{3} \times \frac{\text{sec}}{1.2 \times 10^{-5} \text{ ft}^2} = 2.52 \times 10^5$$

For $e/D = 0.005$, $f = 0.031$ from Fig. 8.14. Using this value, we obtain

$$\bar{V}_2 = \left[2 \times \frac{32.2 \text{ ft}}{\text{sec}^2} \times 80 \text{ ft} \times \frac{1}{0.031(2040 + 8) + 1} \right]^{1/2} = 8.94 \text{ ft/sec}$$

Thus convergence is satisfactory. The volume flow rate is

$$Q = \bar{V}_2 A = \bar{V}_2 \frac{\pi D^2}{4} = \frac{8.94 \text{ ft}}{\text{sec}} \times \frac{\pi \left(\frac{1}{3}\right)^2 \text{ ft}^2}{4} \times \frac{7.48 \text{ gal}}{\text{ft}^3} \times \frac{60 \text{ sec}}{\text{min}}$$

$$Q = 350 \text{ gpm}$$

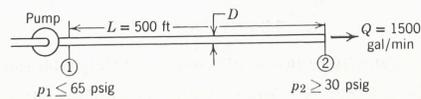
This problem illustrates the procedure for solving pipe flow problems in which the flow rate is unknown. Note that the velocity and, hence, the flow rate, is essentially proportional to $1/\sqrt{f}$. Doubling the value of e/D to account for aging reduced the flow rate by about 10 percent.

Example 8.8

Spray heads in an agricultural spraying system are to be supplied with water through 500 ft of drawn aluminum tubing from an engine-driven pump. In its most efficient operating range, the pump output is 1500 gpm at a discharge pressure not exceeding 65 psig. For satisfactory operation, the sprinklers must operate at 30 psig or higher pressure. Minor losses and elevation changes may be neglected. Determine the smallest standard pipe size that can be used.

EXAMPLE PROBLEM 8.8

GIVEN: Water supply system, as shown.



FIND: Smallest standard D .

SOLUTION:

Δp , L , and Q are known. D is unknown, so iteration will be required to determine the minimum standard diameter that satisfies the pressure drop constraint at the given flow rate.

The maximum allowable pressure drop is

$$\Delta p_{\max} = p_{1 \max} - p_{2 \min} = (65 - 30) \text{ psi} = 35 \text{ psi}$$

Computing equations:

$$\left(\frac{p_1}{\rho} + \alpha_1 \frac{\bar{V}_1^2}{2} + gz_1 \right) - \left(\frac{p_2}{\rho} + \alpha_2 \frac{\bar{V}_2^2}{2} + gz_2 \right) = h_{i_T} \quad (8.28)$$

$$= 0 \quad (3)$$

$$h_{i_T} = h_i + h_m = f \frac{L}{D} \frac{\bar{V}_2^2}{2}$$

- Assumptions: (1) Steady flow
 (2) Incompressible flow
 (3) $h_{i_T} = h_i$, i.e. $h_m = 0$
 (4) $z_1 = z_2$
 (5) $\bar{V}_1 = \bar{V}_2 = \bar{V}$; $\alpha_1 \approx \alpha_2$

Then

$$\Delta p = p_1 - p_2 = f \frac{L}{D} \frac{\rho \bar{V}^2}{2}$$

Since trial values of D are to be assumed, it is convenient to substitute $\bar{V} = Q/A = 4Q/\pi D^2$ so that

$$\Delta p = f \frac{L}{D} \frac{\rho}{2} \left(\frac{4Q}{\pi D^2} \right)^2 = \frac{8fL\rho Q^2}{\pi^2 D^5} \quad (1)$$

The Reynolds number is needed to find f . In terms of Q ,

$$Re = \frac{\rho \bar{V} D}{\mu} = \frac{\bar{V} D}{\nu} = \frac{4Q}{\pi D^2} \frac{D}{\nu} = \frac{4Q}{\pi \nu D}$$

Finally, Q must be converted to cubic feet per second.

$$Q = \frac{1500 \text{ gal}}{\text{min}} \times \frac{\text{min}}{60 \text{ sec}} \times \frac{\text{ft}^3}{7.48 \text{ gal}} = 3.34 \text{ ft}^3/\text{sec}$$

For an initial guess, take nominal 4 in. (4.026 in. i.d.) pipe:

$$Re = \frac{4Q}{\pi \nu D} = \frac{4}{\pi} \times \frac{3.34 \text{ ft}^3}{\text{sec}} \times \frac{\text{sec}}{1.2 \times 10^{-5} \text{ ft}^2} \times \frac{1}{4.026 \text{ in.}} \times \frac{12 \text{ in.}}{\text{ft}} = 1.06 \times 10^6$$

For drawn tubing, $e/D = 0.000,016$ (Fig. 8.15), so $f \approx 0.012$ (Fig. 8.14), and

$$\Delta p = \frac{8fL\rho Q^2}{\pi^2 D^5} = \frac{8}{\pi^2} \times 0.012 \times 500 \text{ ft} \times \frac{1.94 \text{ slug}}{\text{ft}^3} \times \frac{(3.34)^2 \text{ ft}^6}{\text{sec}^2}$$

$$\times \frac{1}{(4.026)^5 \text{ in.}^5} \times \frac{1728 \text{ in.}^3}{\text{ft}^3} \times \frac{\text{lbf} \cdot \text{sec}^2}{\text{slug} \cdot \text{ft}}$$

$$\Delta p = 172 \text{ lbf/in.}^2 > \Delta p_{\max}$$

Since this value is too large, try $D = 6$ in. (actually 6.065 in. i.d.):

$$Re = \frac{4}{\pi} \times \frac{3.34 \text{ ft}^3}{\text{sec}} \times \frac{\text{sec}}{1.2 \times 10^{-5} \text{ ft}^2} \times \frac{1}{6.065 \text{ in.}} \times \frac{12 \text{ in.}}{\text{ft}} = 7.01 \times 10^5$$

For drawn tubing, $e/D = 0.000,010$ (Fig. 8.15), so $f \approx 0.013$ (Fig. 8.14), and

$$\begin{aligned} \Delta p &= \frac{8}{\pi^2} \times 0.013 \times 500 \text{ ft} \times \frac{1.94 \text{ slug}}{\text{ft}^3} \times \frac{(3.34)^2 \text{ ft}^6}{\text{sec}^2} \\ &\quad \times \frac{1}{(6.065)^5 \text{ in.}^5} \times \frac{(12)^3 \text{ in.}^3}{\text{ft}^3} \times \frac{\text{lbf} \cdot \text{sec}^2}{\text{slug} \cdot \text{ft}} \\ \Delta p &= 24.0 \text{ lbf/in.}^2 < \Delta p_{\text{max}} \end{aligned}$$

Since this value is less than the allowable pressure drop, we should check a 5 in. (nominal) pipe. With an actual i.d. of 5.047 in.,

$$Re = \frac{4}{\pi} \times \frac{3.34 \text{ ft}^3}{\text{sec}} \times \frac{\text{sec}}{1.2 \times 10^{-5} \text{ ft}^2} \times \frac{1}{5.047 \text{ in.}} \times \frac{12 \text{ in.}}{\text{ft}} = 8.43 \times 10^5$$

For drawn tubing, $e/D = 0.000,012$ (Fig. 8.15), so $f \approx 0.012$ (Fig. 8.14), and

$$\begin{aligned} \Delta p &= \frac{8}{\pi^2} \times 0.012 \times 500 \text{ ft} \times \frac{1.94 \text{ slug}}{\text{ft}^3} \times \frac{(3.34)^2 \text{ ft}^6}{\text{sec}^2} \\ &\quad \times \frac{1}{(5.047)^5 \text{ in.}^5} \times \frac{(12)^3 \text{ in.}^3}{\text{ft}^3} \times \frac{\text{lbf} \cdot \text{sec}^2}{\text{slug} \cdot \text{ft}} \\ \Delta p &= 55.5 \text{ lbf/in.}^2 > \Delta p_{\text{max}} \end{aligned}$$

Thus the criterion for pressure drop is satisfied for a minimum nominal diameter of 6 in. pipe.

(This problem illustrates the procedure for solving pipe flow problems when the diameter is unknown. Note from Eq. 1 that the pressure drop in turbulent pipe flow is proportional to f/D^5 . The variation of f is small, so Δp at constant flow rate is approximately proportional to $1/D^5$.)

We have solved each of Example Problems 8.7 and 8.8 by direct iteration. Several specialized forms of friction factor versus Reynolds number diagrams have been introduced to solve problems of this type without the need for iteration. For examples of these specialized diagrams, see [16] and [17].

Example Problems 8.9 and 8.10 illustrate the evaluation of minor loss coefficients and the application of a diffuser to reduce exit kinetic energy from a flow system.

Example 8.9

Reference 18 reports results of measurements made to determine entrance losses for flow from a reservoir to a pipe with various degrees of entrance rounding. A copper pipe 10 ft long with 1.5 in. i.d. was used for the tests. The pipe discharged to atmosphere.

View Verification Case 79 Model

[Verification Case 79](#)



Verification Case 80

[View Model](#) [Problem Statement](#)

PRODUCT: AFT Fathom (with ANS Module)

TITLE: FthVerify80.fth

REFERENCE: Michael R. Lindeburg, P.E., Mechanical Engineering Review Manual, 7th Ed., Professional Publications, Example 5.10, page 5-12.

FLUID: Air

ASSUMPTIONS: N/A

RESULTS:

Duct Diameter (inches)	Lindeburg	AFT Fathom
Duct A-B	14	14
Duct C	12	12
Duct D-E	10	10
Duct F	8	8

DISCUSSION:

This duct sizing example used the static regain method to size ducts. The objective of this method is to equalize the pressures along the duct and at each register. This example also placed a maximum velocity limit of 1500 ft/min on the ducts.

To model this in AFT Fathom, two design requirements were created, one with a maximum pressure of 0.4 in H₂O std gauge, and the other with a maximum velocity of 1500 ft/min. The velocity design requirement was applied to all ducts and the pressure design requirement was applied to all ducts that were connected between the registers. The first duct attached to the fan was allowed to exceed the maximum pressure design requirement.

A special material model was created of circular ducts from 4 inches to 36 inches with the inner diameter equal to the nominal size.

The automated sizing was discrete with the objective to minimize flow volume.

[List of All Verification Models](#)

Verification Case 80 Problem Statement

Verification Case 80

Michael R. Lindeburg, P.E., Mechanical Engineering Review Manual, 7th Ed., Professional Publications, Example 5.10, page 5-12.

Lindeburg's Title Page

5-12

FANS AND DUCTWORK

step 3: Assume a main run velocity, v_m , from table 5.5.

step 4: Size the main run from $A_m = \frac{Q_m}{v_m}$.

step 5: Find the equivalent length of duct from the fan to the first take-off. It may be necessary to make some assumptions about the bend radii.

step 6: From figure 5.4, find the friction loss in the main run up to the first take-off. This is Δp_{main} .

step 7: The design pressure at the nearest grille outlet is p_d . Therefore, the pressure at the start of the main run is

$$p_{s, \text{start}} = p_d + \Delta p_{main} \quad 5.42$$

step 8: If the fan outlet and the main velocities are different, calculate the regain.

$$\Delta p_{fan} = R \left(\left(\frac{v_{fan}}{4005} \right)^2 - \left(\frac{v_m}{4005} \right)^2 \right) \quad 5.43$$

If v_m is larger than v_{fan} , use $R = 1.1$. Otherwise, use $R = .75$.

step 9: Find the required fan outlet pressure.

$$p_{s, \text{fan}} = p_{s, \text{start}} - \Delta p_{fan} \quad 5.44$$

If $p_{s, \text{fan}}$ is too high, choose a lower v_m and go to step 4.

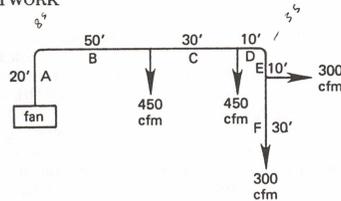
step 10: Find the equivalent length between grilles and take-offs. It may be necessary to make assumptions about sizes and radii. Proportion p_{var} along the grille run, giving the higher losses to longer runs and to runs with fittings.

step 11: Knowing L_e and Q_2 for each section, find v_2 from figure 5.6.

step 12: Solve for the duct size from $A = \frac{Q}{v}$.

Example 5.10

Use the static-regain method to size the duct system shown. The fan outlet is 1500 cfm at 1700 fpm. The equivalent length of the bend is 15 feet.



step 1: Assume $p_{var} = .10''$ w.g.

step 2: Assume an average $p_d = .25''$ w.g.

step 3: Assume $v_m = 1500$ fpm.

step 4: $A_m = \frac{1500}{1500} = 1$ sq. ft. For a round duct, the diameter will be approximately 14''.

step 5: The equivalent length of sections A and B, including the bend, is

$$20 + 15 + 50 = 85'$$

step 6: From figure 5.4, the friction loss per 100 feet is $.20''$ w.g. Thus, the loss in sections A and B is

$$\left(\frac{85}{100} \right) .20 = .17'' \text{ w.g.}$$

step 7: The pressure at outlet 1 is $.25''$ w.g. The pressure at the start of run A is $.25 + .17 = .42''$ w.g.

step 8: The fan regain is

$$.75 \left[\left(\frac{1700}{4005} \right)^2 - \left(\frac{1500}{4005} \right)^2 \right] = .03'' \text{ w.g.}$$

step 9: The required fan pressure is $.42 - .03 = .39''$ w.g.

step 10: The equivalent lengths of the remaining sections are found:

section	equivalent length	Q
C	30	1050
D/E	10 + 10 + 15 = 35	600
F	30	300
	<u>115' total</u>	

p_{var} is distributed along CDEF in proportion to the equivalent length. Thus, $.03$ is distributed to sections C and F, and $\frac{p}{L}$ is given as $.04''$ w.g.

step 11: $L_e/(Q^{.61})$ is calculated for each section:

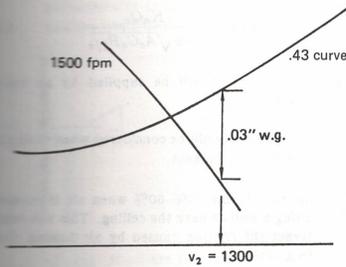
section	$L_e/Q^{.61}$
C	0.43
D/E	0.71
F	0.92

v_2 is found from figure 5.6, allowing for a loss in the static pressure between the v_1 and $L_e/Q^{.61}$ curves equal to the proportion of P_{var} calculated in step 10.

$$v_{2,C} = 1275 \text{ fpm}$$

$$v_{2,D/E} = 1050 \text{ fpm}$$

$$v_{2,F} = 850 \text{ fpm}$$



step 12: The required duct areas and diameters are:

$$A_C = \frac{1050}{1275} = 0.82 \quad \text{or } 12'' \text{ diameter}$$

$$A_{D/E} = \frac{600}{1050} = 0.57 \quad \text{or } 10'' \text{ diameter}$$

$$A_F = \frac{300}{850} = 0.35 \quad \text{or } 8'' \text{ diameter}$$

5 DAMPERS

Dampers are included in all runs to balance the system. Balancing is the act of closing down dampers to equalize the friction losses in all ducts. Without dampening, the majority of the air flowing would escape out the closest grille.

It is generally a good idea to install dampers even when more sophisticated design methods are used. Such dampers can be used for fine-tuning the installation.

Dampers can be motorized or operated manually. Figure 5.7 illustrates several common types of dampers.

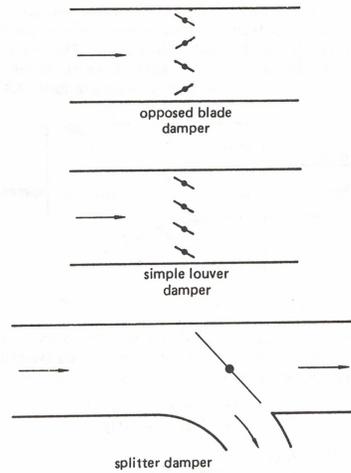


Figure 5.7 Types of Dampers

It is possible to design the outlet grilles to act as a damper. Such registers may be designed as a plate with multiple small holes through which the air must escape. The disadvantage of combining the tasks of air distribution and friction generation is that the noise created by the friction generation is projected directly into the room. It is better to have dampers installed close to the main supply and as far away from the grille as possible.

6 AIR DISTRIBUTION

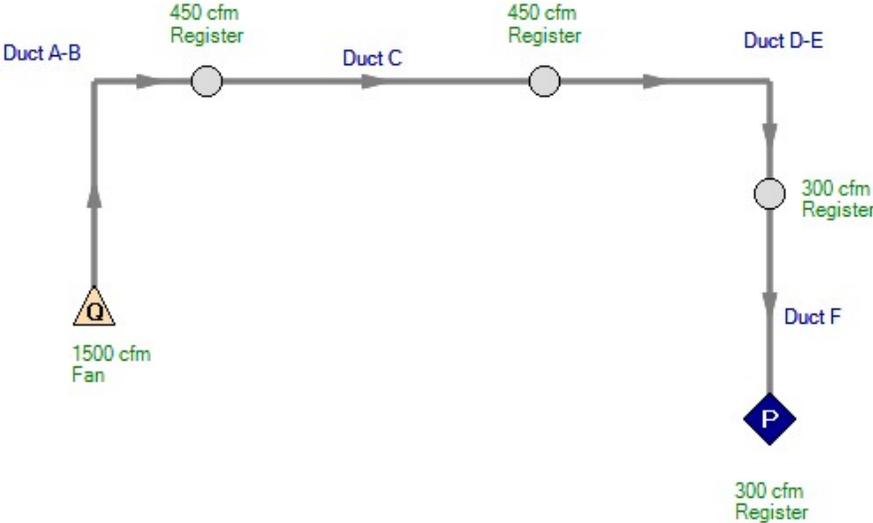
An *outlet* is the general term used to describe any opening through which air enters the treated space. Although they are not strictly adhered to, the following definitions can be used to distinguish among outlets with different functions.

- grille: a decorative covering for an outlet
- diffuser: a functional grille guiding air direction
- register: a grille with an internal damper

The *gross area* or *core area* of the grille is its total cross sectional area. The total area of the openings in the grille constitutes the *free area* or *daylight area*.

View Verification Case 80 Model

[Verification Case 80](#)



Index

#

2D Freeform 367

4-Component 136, 138, 558, 568, 579, 585

A

About AFT Fathom 369

Absolute roughness 217

Action if Setpoint not Achievable 279

Activate Add-on Modules 380

Activate Modules 365, 374

Add Component to Library 365

Add/Remove Segments 367

Additional (Minor) Losses 218

Additional Losses 218

Add-on Module Capabilities 28

Adiabatic (Perfectly Insulated) 220, 223

AFT Blog 369

AFT Home Page 368

AFT Notifications 369

AFT Standard fluid 91

Align 367

Allowable Operating Region (AOR) 299

Always Control (Never Fail) 280

Ambient Conditions 224

Ambient Pressure 169

AOR 299
Area Change 269
ASME Steam/Water 91
Assigned Flow 271
Assigned Pressure 272
Atmospheric Pressure 169
Auto-rotate Icon 367

B

Backup Settings 454
Balance Energy At Junction 78
Base Scenario 498
Bend 272
BEP 298
Best Efficiency Point (BEP) 298
Bingham Plastic 115
Bookmarks 363
Boundary Conditions 755
Bounded by known flow rates 72
Branch 277
Buried Pipe 221

C

Cautions 772
Check Valve 277
Check Valve at Discharge (No Backflow Allowed) 295
Checklist 374

Chempak 95-96

Chempak Fluid 91

Clear Initial Guesses 362

Closed Tank 609

Color Map 392

Colors and Fonts 451

Command Buttons 423

Compare Scenarios 364

Conductivity/Convection Data Source 225

Connectivity Requirements 207

Constant Heat Flux 221, 223

Constant Heat Rate 221, 223

Control Valve 278

Control.aft File 816

Controlled Downstream Temperature 284

Convective Heat Transfer 220

Convective Heat Transfer and Heat Flux 221, 224

Convective Heat Transfer with Heat Tracing 221, 224

Conventions 69, 257

Converting models from earlier versions 817

Copy 361

Copy Graphics 361

Copy X-Y Data 361

Cost 535

Cost Application Manager 365

Cost Libraries 524

Cost Report 171, 419, 517
Cost Report Formatting 421
Cost Summary 365
Cost Time Period 172
Counter-Flow 284
Critical Warnings 772
Cross Flow, Both Fluids Unmixed 284
Cross Flow, Secondary Fluid Mixed 284
Cross Flow, System Fluid Mixed 284
Cross-Plotting Features 429
Custom Libraries 707, 713-714
Custom Units 365
Customize Icon 362
Cut 361

D

Dead End 281
Delete 361
Deposition Velocity 139
derating 577
de-rating 577
Design Alerts 364
Design Factor 239, 256
Design Factors 247, 256
Disable Pipes Snapping to Isometric Grid 367
Display Parameters 419
Dittus-Boelter 235

Duplicate 361

Duplicate Special 362

E

eLicense Information 365

End Of Curve Flow Rate 299

Energy Balance 335

Energy Cost 172

Energy Cost Library 524

Enthalpy Stagnation Rise 284

ESRI 462

Exit 361

Exit Valves 351

Explicit Friction factor 217

Export Model Data 360

Export Output 360

Extended Model Check 363

External Heat Flux 224

F

Favorite Fittings & Losses 220

FCV 278

Fei Xiangjun 140, 565-566

File Locations 454

Find 363

Fire Protection 175

Fittings & Losses 365

Fittings and Losses 218
Flip Vertex 367
Flow Control Valves 278
Flow Expert Package 368
Fluid Library 365
Fluid Property Viewers 365
Format & Action 420
Friction Model 216
Frictionless 217

G

General Component 282
Generate Disconnected Scenario 364
Global Junction Edit 362
Global Pipe Edit 362
Gnielinski 235
Goal Seek & Control 363
Graph List Manager 427
Graph Outline Colors 428
Graph Results 368, 372
Graph Results Toolbar 427
Graph Tab Actions 429
Graph Tab Icons 429
Grid 449
Groups 362
GSC Module 545, 549

H

Hanks' Bingham Plastic Friction Model 140, 565
Hazen-Williams 218
Hazen-Williams Method 244
Heat Exchanger 282
Heat Exchanger Summary 419
Heat Tracing on Pipe (Flux) 224
Heat Transfer 121, 220, 231, 283
Heat Transfer Panel 121
Helical Tubes 244
Highlight in Pipe and Jct Windows 363
HT Area Ratio 225
Hydraulically smooth 217

I

Import Excel Change Data 360
Import Graph Data 360
Infinite source or sink of fluid 73
Information Messages 772
Initial Flow Guess 238
Initial Guesses 255
Input 408
Inspection 209
Insulation Library 365
Intelliquip 308
Intermediate Pipe Elevations 239

Interpolated X-Y Data 79

Interstage Bleed 295

Isometric 367

Isothermal 220, 223

J

Jet Pump 287

Join Selected Pipes/Jcts 362

Junction Deltas 421

Junction Elevation 255

Junction library 365

Junction Number 254

Junctions 203

 Area Change Properties Window 269

 Assigned Flow Properties Window 271

 Assigned Pressure Properties Window 272

 Bend Properties Window 272

 Branch Properties Window 277

 Check Valve Properties Window 277

 Control Valve Properties Window 278

 Dead End Properties Window 281

 General Component Properties Window 282

 Heat Exchanger Properties Window 282

 Jet Pump Properties Window 287

 Orifice Properties Window 287

 Pump Properties Window 291

 Relief Valve Properties Window 330

Reservoir Properties Window 333
Screen Properties Window 335
Spray Discharge Properties Window 337
Tee/Wye Properties Window 341
Three Way Valve Properties Window 349
Valve Properties Window 350
Venturi Properties Window 352
Volume Balance Properties Window 353
Weir Properties Window 355

K

Keep Network eLicense Checked Out 365
Keyboard Shortcuts 368
Known Parameters Initially 603

L

Labels 390, 450
Laminar Flow 126, 243
Last Scenario 364
Last View 363
Libraries 707
Library Connections 423
Library Manager 366
List Undefined Objects 363
Liu Dezhong 140, 562-565
Load Background Picture 361
Lock Object 367

Loss factor 257

Loss Model 338

M

Make Selected Pipes and Jcts Orthogonal 367

Merge 360

Miller Turbulent 218

Miller Turbulent Method 246

Minimizing the Quick Access Panel 381

MIT Equation 218

MIT Equation for Crude Oil 245

Model Data 368, 372

Model Data Control 364

Model Data Print Content 364

Model Diagnostics 365

Model Status Light 380

Modeling a Constant Wall Temperature 226

Multi-Scenario 422

N

National Fire Protection Association 175

New 360

NFPA 175

NFPA Configuration 364

NIST REFPROP 91

NPSH Reference Elevation 291

NPSHR Affinity Exponent At Operating Speed 299

NTU-Effectiveness Models 284
Number of Insulation Layers 224

O

Object Status 207
Open 360
Open Percentage Data 351
Open Percentage Table 279
Orifice 287
Orthogonal 367
Output 368, 372
Output Control 364
Output File 815
Output Language 364
Output Messages 772
Output Print Content 364
Output Window Updates 415
Overview Map 363, 379

P

Parallel Pipes 239
Particle size distribution 561
Particle Size Distribution 576
Paste 361
PDCV 278
Pipe Connectivity 254
Pipe Diameter 217

Pipe Drawing Mode 367
Pipe Elevations 239
Pipe Geometry 216
Pipe Heat Transfer Output 419
Pipe ID Numbers 198
Pipe Line Size and Color 389
Pipe Material 216
Pipe Material Library 365
Pipe Segments 198
Pipes 203, 218, 224, 241
Polynomial 79
POR 299
Power Law 115
Preferred Operating Region (POR) 299
Preferred Units 447
Preformatted Report 360
Pressure Drop 218
Pressure Drop Control Valves 278
Pressure Reducing Valves 278
Pressure Sustaining Valves 278
Primary Windows 366, 369, 372
Print Preview 360
Printing and Exporting 417, 431
Profile 426
Properties Tab 375
PRV 278

PSV 278

Pump 291, 309

 Intelliquip 308

Pump Data 297

Pump Sizing with a Flow Control Valve 308

Pump Summary 419

Q

Quick Access Panel 386

Quick Graph Formatting 364

Quick Start Guide 368

Quick Start Video 368

R

Recovery File 455

Reference pressure 72, 75

Relative roughness 217

Relief Valve 330

Remote Sensing 332

Renumber 362

Renumber Automatic 399

Renumber Increment 399

Renumber Wizard 399

Reservoir 333

Reservoir Summary 419

Reset Label Locations 367

Resistance Geometry 224

Restricted Area 288, 351
Results Diagrams 415
Reverse Direction 367
Rotate Icon Right or Left 366
Rotate Selection Right or Left 367
Round-Edged Orifice 289
Ruby & Zanke (1977) 140, 563

S

Save 360
Save As 360
Save Graph As 360
Save Graph Data As 360
Scale Tables 520
Scale/Flip Workspace 367
Scenario 508
Scenario Comparison Tool 364
Scenario Manager 364, 374
Screen 335
Search 184
Select All 362
Select Flow Path 362
Select Graph Parameters 364
Select Last Selection 362
Select Objects 363
Select Special 362
Set Pipe and Junction Locations to Parent 367

Settling Slurry 551
Shell & Tube, 1 Shell Pass, Mult. of 2 Tube Passes 284
Show Examples 368
Show Grid 367
Show Object Status 363
Show/Hide Unit Items 448
Slurry 551
Slurry System Curve 429
Snap to Grid 367
Sort Table By 366
Special Conditions 239, 256, 362
Specified Heat Rate In Constant 284
Specified Heat Rate In vs. Flow 284
Specify the required property data 206
Split Pipe at Intermediate Elevations 362
Spray Discharge 337
Start Batch Run 360
Status Feature 207
Submerged Pump 294

T

Takeoff Flow 295
Technical Seminars 368
Tee/Wye 341
Temperature Stagnation Rise 284
Thermal Link 285
Thermal Resistance 233

Thermal Results in Output 221
Thomas (1965) 140, 565
Three Way Valve 349
Time Simulation Formatting 421
Transfer Results to Initial Guesses 362
Transient 586
Troubleshooting 803-804
Troubleshooting Models 801
Turbulent Flow 243

U

Undefined Objects Window 207
Undo 361
Unique solution 72
User Library Fluid 91
User Options 365, 447, 449, 452, 454

V

Valve 350
Valve Summary 419
Venturi 352
Video Tutorials 368
Viscosity Model 115
Volume Balance 353
vortexing 490

W

Wall Temperature 226

Warnings 772

Wasp-Durand 138, 140, 561, 564, 569

Weir 355

What Can AFT Fathom Do 27

Wilson, Addie, Clift 134, 137-139, 552-555, 568, 585

Workspace 368, 372, 385

Workspace Icon 256

Workspace Layers 385

 Active Layer 388

 From Workspace Selections 387, 923

Workspace Layers Tab 378

Workspace Pan 363

Workspace Symbols 394, 452