

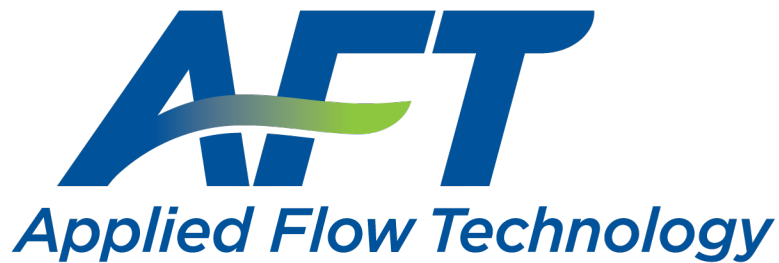
# ***AFT Arrow***<sup>™</sup>

## **Examples - English Units**

**AFT Arrow Version 9**

**Compressible Pipe Flow Modeling**

Published: October 11, 2023



*Dynamic solutions for a fluid world* <sup>™</sup>

# Contents

---

<b>Walk-Through Examples</b> .....	<b>4</b>
Beginner: Air Heating System .....	6
Flow Through an Orifice .....	33
Control Valve .....	38
Natural Gas Burner .....	45
Compressed Air System .....	52
Process Steam System .....	65
Hydrocarbon Relief System .....	72
Refinery Relief System .....	79
Air Blower Sizing .....	89
Compressed Air System - Cost .....	98
Process Steam System - Cost .....	107
<b>Module Walk-Through Examples</b> .....	<b>124</b>
Goal Seek and Control (GSC) Examples .....	125
Beginner: Heat Transfer in a Pipe - GSC .....	125
Steam Relief System - GSC .....	134
Natural Gas Burner - GSC .....	138
Compressed Air System - GSC .....	144
Process Steam System - GSC .....	151
Automatic Network Sizing (ANS) Examples .....	157
Beginner: Three Tank Steam System - ANS .....	158
Control Valve - ANS .....	172
Air Distribution, Creating Libraries - ANS .....	178
Air Distribution - ANS .....	193

---

Air Distribution, Multiple Design Cases - ANS .....	218
Helium Transfer - ANS .....	227
Natural Gas Burner - ANS .....	237
Process Steam System - ANS .....	253
Extended Time Simulation (XTS) Examples .....	261
Air Receiver Tank - XTS .....	263
<b>Index .....</b>	<b>275</b>

## Walk-Through Examples

The AFT Arrow Examples Help System is designed to expose users to a range of applications and improve their understanding of the AFT Arrow software. If you are new to AFT Arrow, start with the Air Heating System Example, because it has the most in-depth discussion on how to create and run AFT Arrow models. After this example, choose an example from the table below that most closely matches your application.

Example	Complexity	Fluid	Pipes	Fans / Compressors	Description
<a href="#">Beginner: Air Heating System</a>	Beginner	Air	5	1	A new AFT Arrow user should start here. Get in-depth instructions of model building by solving an air heating system 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.
<a href="#">Flow Through an Orifice</a>	Beginner	Steam	2	0	Learn about the CdA value for sonic choking in the Orifice Properties window. See how the effective flow area affects the flow rates and pressures in the system.
<a href="#">Control Valve</a>	Beginner	Steam	2	0	Learn how to define the Convective Heat Transfer model on the Heat Transfer tab of the Pipe Properties window. See how to use a control valve junction to determine pipe size. Learn how to use the Output Control window to display relevant heat transfer output parameters.

## Walk-Through Examples

Example	Complexity	Fluid	Pipes	Fans / Compressors	Description
<a href="#">Natural Gas Burner</a>	Beginner	Methane	16	0	Learn how to use the Specified Heat Rate In Constant thermal model in a Heat Exchanger junction to obtain a desired outlet temperature in a system. See how to specify a resistance curve as the loss model in a Heat Exchanger junction.
<a href="#">Compressed Air System</a>	Beginner	Air	9	1	Learn how to input a compressor curve into a Compressor/Fan junction. See how to use the Scenario Manager to create different system configurations. Explore how to use the isometric pipe drawing mode in the Workspace window.
<a href="#">Process Steam System</a>	Intermediate	Steam	17	0	Learn about the Solution Method panel in Analysis Setup to see how the Length March Increment impacts results.
<a href="#">Hydrocarbon Relief System</a>	Intermediate	Methane Hydrogen	4	0	Learn how to create a mixture using the NIST REFPROP fluid library. See how and where multiple choking points occur in the system.
<a href="#">Refinery Relief System</a>	Beginner	Methane Ethane Propane	6	0	Learn how to use the NIST REFPROP fluid library to specify multiple pure fluids in a single system where dynamic mixing occurs. See how to use Global Junction Edit to make rapid changes to junctions.
<a href="#">Air Blower Sizing</a>	Intermediate	Air	13	1	Learn how blower efficiency affects pressure rise and discharge temperature. See how using heat transfer further impacts the pressure and temperature results.

## Walk-Through Examples

Example	Complexity	Fluid	Pipes	Fans / Compressors	Description
<a href="#">Compressed Air System - Cost</a>	Beginner	Air	9	1	Learn the basics of cost calculations such as specifying material, installation, and energy costs, connecting engineering and cost libraries, and interpreting the Cost Report in the Output window.
<a href="#">Process Steam System - Cost</a>	Intermediate	Steam	17	0	Learn how to create cost libraries for pipe materials and junctions. See how to use Scale Tables to allow costs to scale as pipe diameters change. Learn how to specify Material costs versus Installation costs.

### Beginner: Air Heating System

#### Summary

This example is designed to demonstrate the big picture of AFT Arrow's layout and structure. Some of the more basic concepts will be used to build a simple air heating system driven by a compressor.

#### Topics Covered

- Completing Analysis Setup
- Drawing the system with Pipes and Junctions
- Entering Pipe and Junction properties

#### Required Knowledge

This example assumes the user has not used AFT Arrow previously. It begins with the most basic elements of laying out the pipes and junctions and solving hydraulic methodology.

#### Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Arrow installation:

- ***US - Air Heating System.aro***

### **Step 1. Start AFT Arrow**

---

➤ **To start AFT Arrow**, click Start on the Windows taskbar, choose the AFT Arrow 9 folder, then click AFT Arrow 9 to launch the application. (This refers to the standard menu items created by setup. You may have chosen to specify a different menu item during installation.)

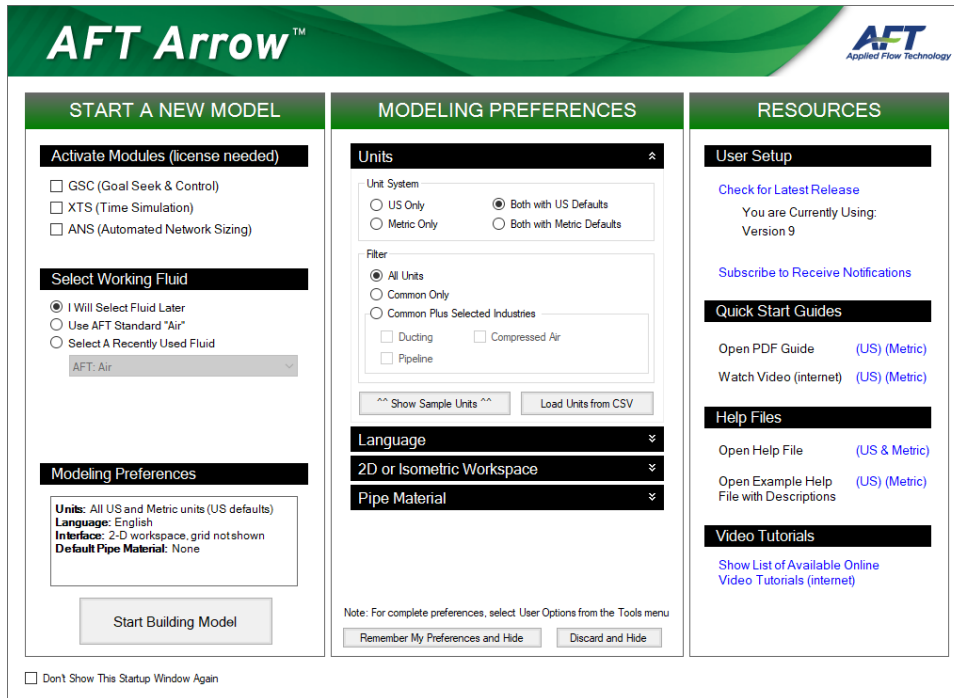
After AFT Arrow opens, the Start-Up screen, as shown in Figure 1, appears and provides you with several options before you start building a model. Some of the actions available are:

- Open a recent model, browse to a model, or browse to an Example
- Activate an Add-on Module
- Select AFT Standard Air 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 sets and check the box to Set as my Default.

---



**Figure 1: AFT Arrow Start-Up window**

Click Start Building Model on the Start-Up window, the Workspace window is always the active (large) window, as seen in Figure 2. The five tabs in the AFT Arrow window represent the five Primary Windows. Each Primary Window contains its own Toolbar that is displayed directly beneath the Primary Window tabs.

AFT Arrow supports dual monitor usage. You can click and drag any of the five Primary Window tabs off of the main AFT Arrow window. Once you drag one of the Primary Windows off of the AFT Arrow 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 AFT Arrow 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 Arrow 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 far right. It is possible to minimize the Quick Access Panel by clicking on the thumbtack pin in the upper right corner of the Quick Access Panel in order to allow for greater Workspace area.

The Workspace takes up the rest of the window.

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 the Annotation tool. The Pipe Drawing tool, on the upper left corner of the



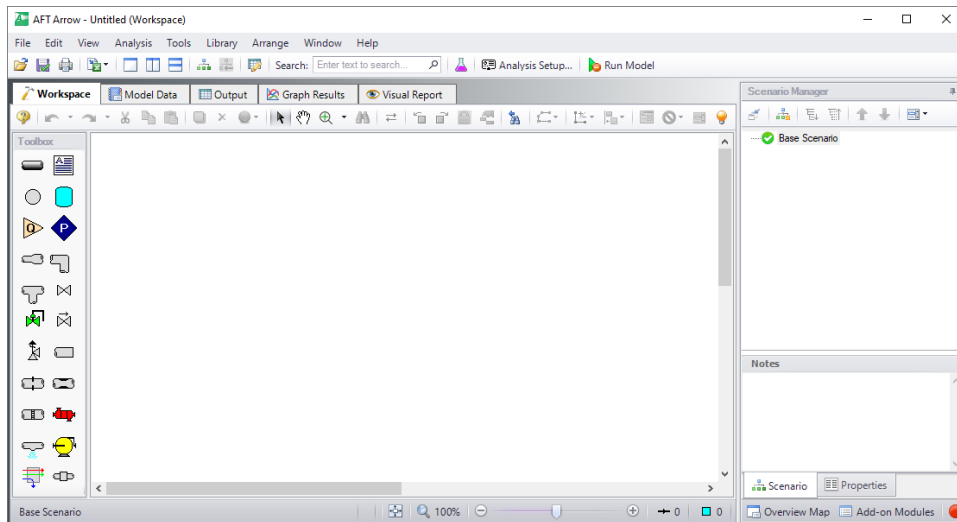
## Walk-Through Examples

---

Toolbox, 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 icons that represent the different types of junctions available in AFT Arrow. Junctions are components that connect pipes and also influence the pressure or flow behavior of the pipe system. The twenty junction icons can be dragged from the Toolbox and dropped onto the Workspace.

When you hold your mouse pointer over any of the Toolbox tools, a ToolTip identifies the tool's function.



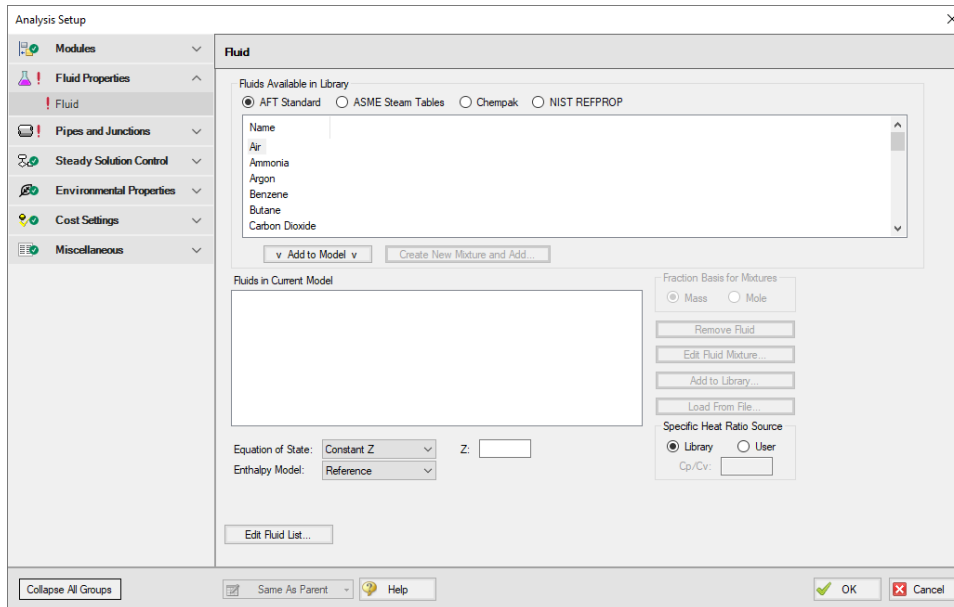
**Figure 2: The Workspace window is where the model is built**

### Step 2. Complete the Analysis Setup

---

➤ **Next, click Analysis Setup on the Toolbar** that runs across the top of the AFT Arrow 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 before AFT Arrow allows you to run the Solver.

Analysis Setup can also be accessed by clicking on the Status Light on the [Quick Access Panel](#). Once Analysis Setup is complete, the Model Status Light then 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

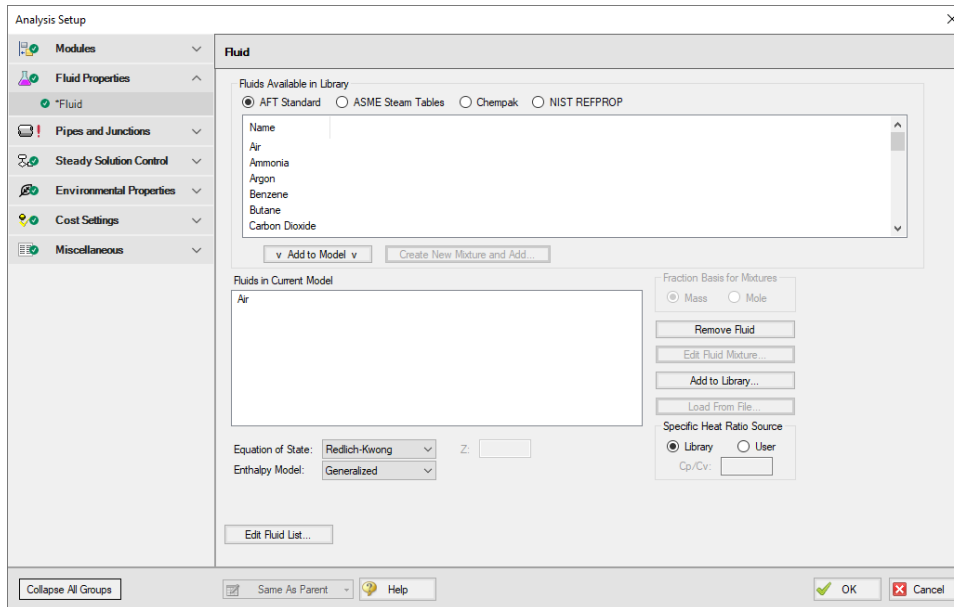
Although Analysis Setup initially opens to the Fluid panel of the Fluid Properties group, the first group, Modules, always has a green check when you start AFT Arrow because there are no modules activated by default. No further input is required here.

## B. Define the Fluid Properties Group

Next is the Fluid Properties group. This group allows you to choose a fluid, select an Equation of State, and select an Enthalpy Model. Start by clicking on Fluid to open the Fluid panel (see Figure 4).

You can select a fluid from the standard AFT Arrow fluid library (AFT Standard), select ASME Steam Tables, or select multiple fluids and/or create mixtures using the Chempak library or NIST REFPROP library. You can also select a custom fluid, which will be shown in the AFT Standard list. Custom fluids are created by opening the Library Manager from the AFT Arrow Library menu or by clicking the Edit Fluid List button at the bottom of the Fluid panel and clicking Add New Fluid...

➤ **Select AFT Standard, select Air, then click Add to Model (see Figure 4).** Leave the Equation of State model as Redlich-Kwong and the Enthalpy Model as Generalized.



**Figure 4: The Fluid panel lets you select gases for the model and create mixtures**

### 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 recirculating model, you will place five junctions on the Workspace. Then you will connect the junctions with pipes.

#### I. Place the tank

➤ **To start, drag a tank junction from the Toolbox and drop it on the Workspace.** Figure 5 shows the Workspace with one tank.

#### **Objects and ID numbers**

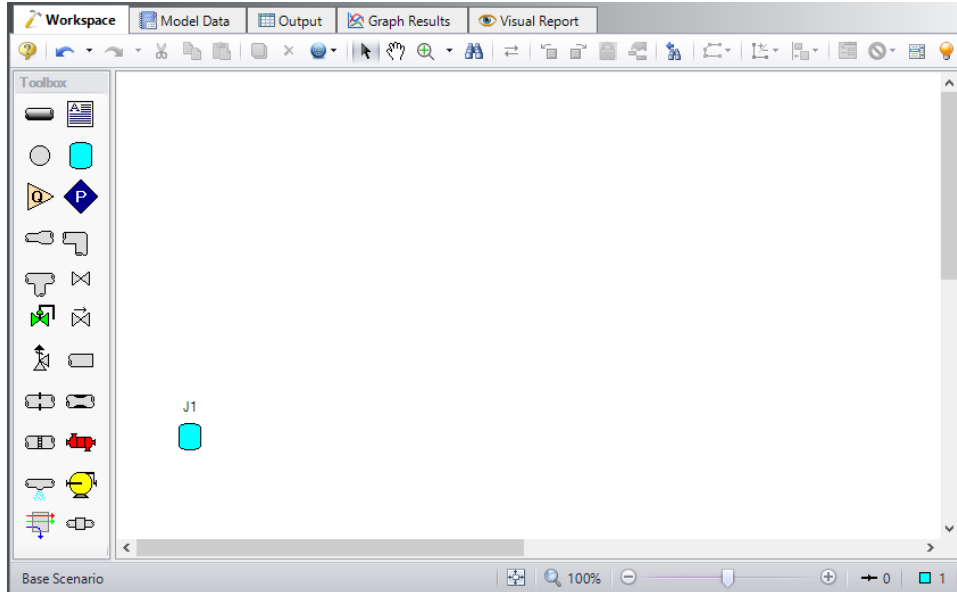
Items placed on the Workspace are called *objects*. All objects are derived directly or indirectly from the Toolbox. AFT Arrow uses three types of objects: *pipes*, *junctions* and *annotations*.

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 tank you have created 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. Duplicating, cutting, copying, and pasting are all supported.



**Figure 5: Adding a tank junction to the model**

## II. Place the other junctions

If the new junction type you want to add already exists on the Workspace, you have the option of duplicating that junction. You do this by selecting the object on the Workspace and then either choosing Duplicate from the Edit menu or pressing Control+D on the keyboard.

Drag a compressor/fan junction from the Toolbox and drop it somewhere to the right of the tank (see Figure 6). Next drag a branch junction and locate it above the compressor, then drag a heat exchanger junction and locate it above the tank. Finally, drag another tank junction and place it above the first tank junction.

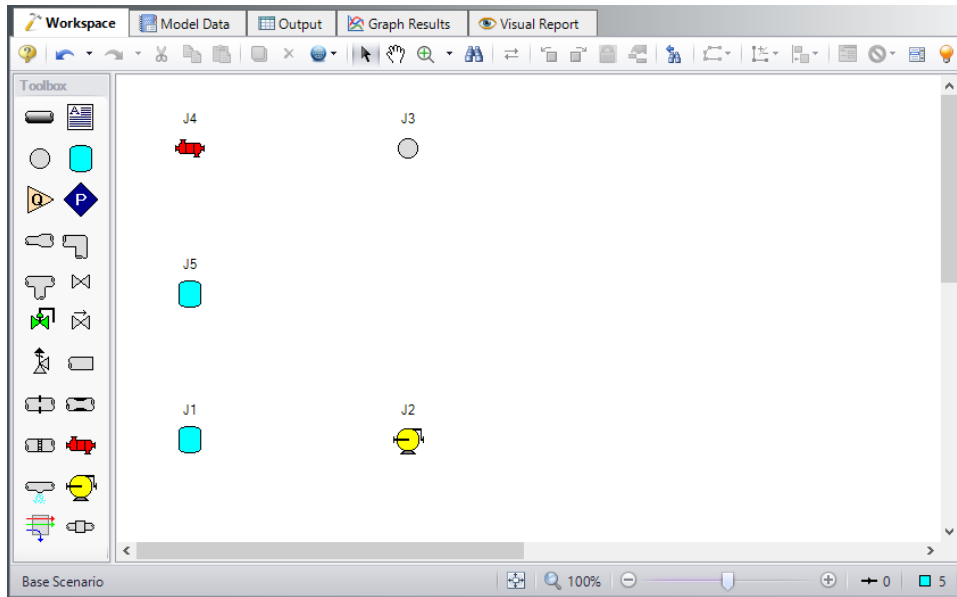
---

**Note:** The relative location of objects in AFT Arrow is not important. Distances and heights are defined through Pipe and Junction Property windows. The relative locations on the Workspace establish the connectivity of the objects, but have *no bearing on the actual length or elevation relationships*.

---

If the icons do not line up exactly, no problems will result. However, your model may have a nicer appearance if the icons line up. You can align the icons by using the align features selected from the Arrange menu.

Your model should now appear similar to that shown in Figure 6.



**Figure 6: Adding additional junctions to the model**

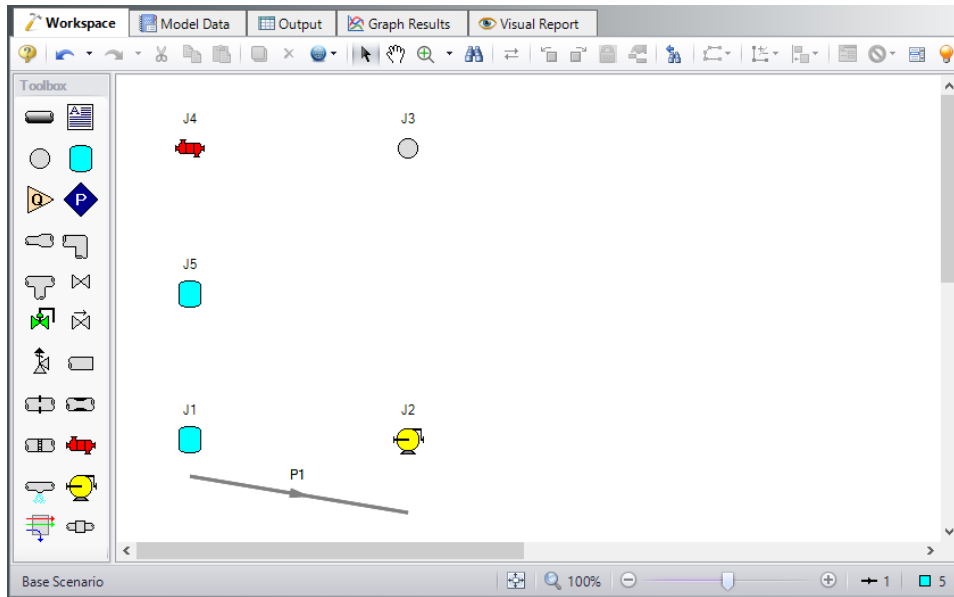
➤ **Before continuing, save the work** you have done so far. Choose Save As... from the File menu and enter the file name Air Heating System and AFT Arrow will append the \*.aro extension to the file name.

### III. Draw a pipe between J1 and J2

Now that you have five junctions, you need to connect them with pipes.

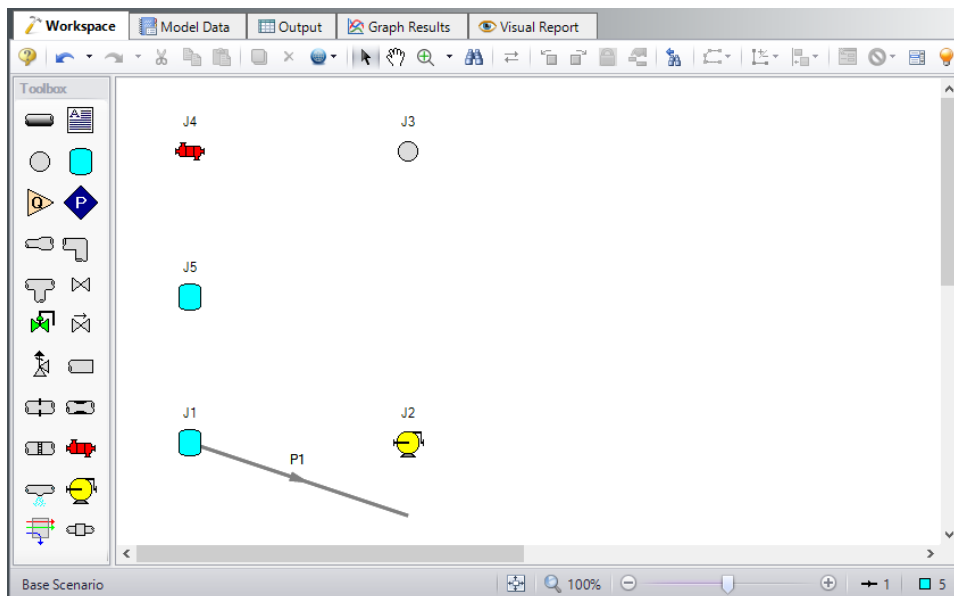
➤ **To create a pipe**, click the Pipe Drawing Tool icon on the Toolbox. The pointer will change to a cross hair when you move it over the Workspace. Draw a pipe below the junctions, similar to that shown in Figure 7.

The pipe object on the Workspace has an ID number (P1) shown near the center of the pipe.



**Figure 7: Create a Pipe**

➤ **To place the pipe** between J1 and J2, use the mouse to grab the pipe in the center, drag it so that its left endpoint falls within the J1 Tank junction, then drop it there (see Figure 8). Next, grab right end-point of the pipe and stretch the pipe, dragging it until the endpoint terminates within the J2 Compressor/Fan junction (see Figure 9).



**Figure 8: Connecting pipe P1 to the tank J1**

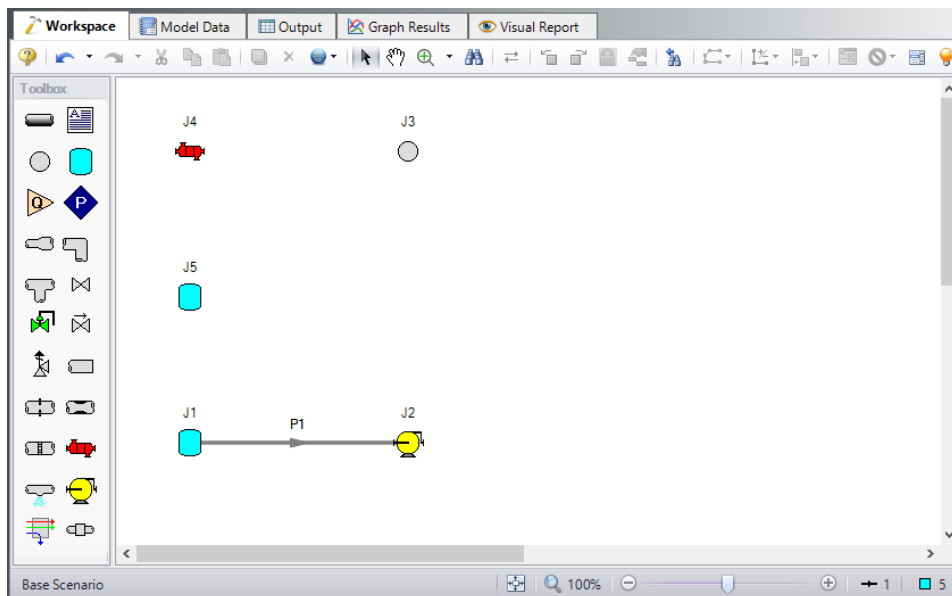
### **Reference positive flow direction**

On each pipe is an arrow that indicates the reference positive flow direction for the pipe. AFT Arrow 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 Pipe Direction button on the Workspace Toolbar.

The reference positive flow direction is used for reference purposes only and need not be the actual flow direction.

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.



**Figure 9: Connecting pipe P1 between two junctions**

## **IV. Draw the remaining pipes**

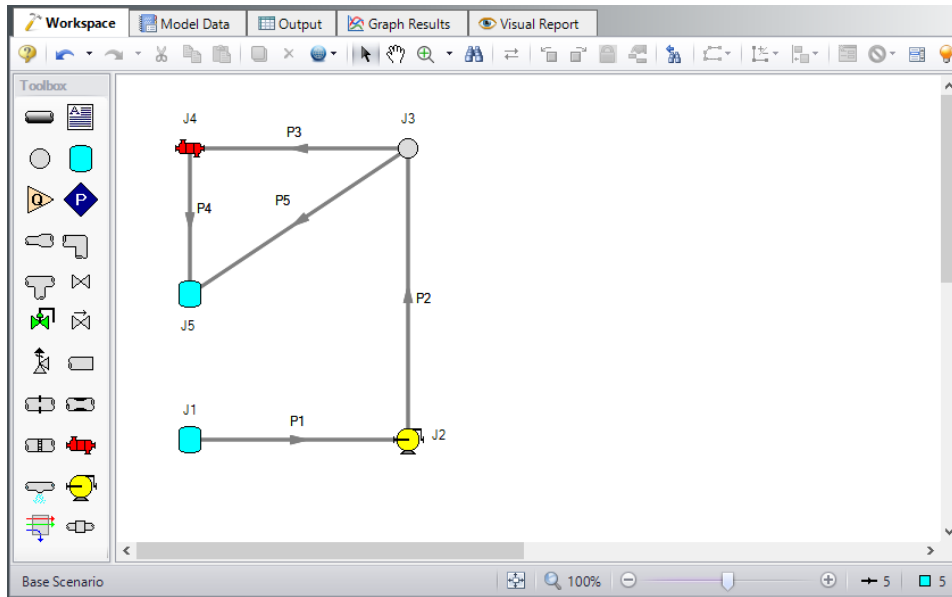
A faster way to add a pipe is to draw it directly between the desired junctions.

➤ **Activate the pipe drawing tool** again, position the mouse pointer on the J2 compressor/fan, then press and hold the left mouse button. Stretch the pipe up to the J3 Branch then release the mouse button. Now draw a third pipe from the J3 Branch to the J4 Heat Exchanger, draw a fourth pipe from the J4 Heat Exchanger to the J5 Tank, and draw a fifth pipe from the J3 Branch to the J5 Tank. Your model should now look similar to Figure 10.

---

**Tip:** Double-click the Pipe Drawing Tool in order to draw multiple pipes at a time, then deactivate the Pipe Drawing Tool either by pressing the Esc key or single-clicking the Pipe Drawing Tool again.

---



**Figure 10: Workspace view after drawing remaining pipes**

At this point all the objects in the model are graphically connected. Save the model by selecting Save in the File menu or by clicking on the diskette button on the Toolbar.

**Note:** Some users find it desirable to lock 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 Workspace 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 in the User Options window from the Tools menu and specify that the pipes and junctions snap to grid.

## V. 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 Arrow's requirements. To see the status of the objects in your model, click the flood light (Show Object Status) icon on the far right of the Workspace Toolbar (alternatively, you could choose Show Object Status from the View menu). Each time you click the light, 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 the User Options window from the Tools menu.)



Because you have not yet defined the pipes and junctions in this example, 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.

## **VI. Define Tank J1**

**To define the first tank**, open the J1 Tank Properties window by double-clicking on the J1 icon. The table on the Loss Coefficients should show you that Pipe 1 is connected to this tank.

---

**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 Workspace Toolbar.

---

➤ **Enter the Pressure as 0 psig.** You can assign any unit of pressure found in the adjacent drop-down list box of units. Enter the Temperature as 100 deg. F and the Elevation as 0 feet.

---

**Note:** You can choose default units for many parameters (such as meters for length) in the User Options window.

---

You can give the component a name, if desired, by entering it in the Name field at the top of the window. By default, the junction's name indicates the junction type. The name can be displayed on the Workspace, Visual Report 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 Jct list. The current junction will get the input properties from the library.

The Copy Data From Junction... list will show all the junctions of the same type in the model. This will copy selected parameters from an existing junction in the model to the current junction.

### ***Using the folder tabs in the Properties windows***

The information in the Properties windows is grouped into several categories and are placed on separate tabs. Click the folder tab to bring its information forward. Figure 11 is an example of a Properties window.

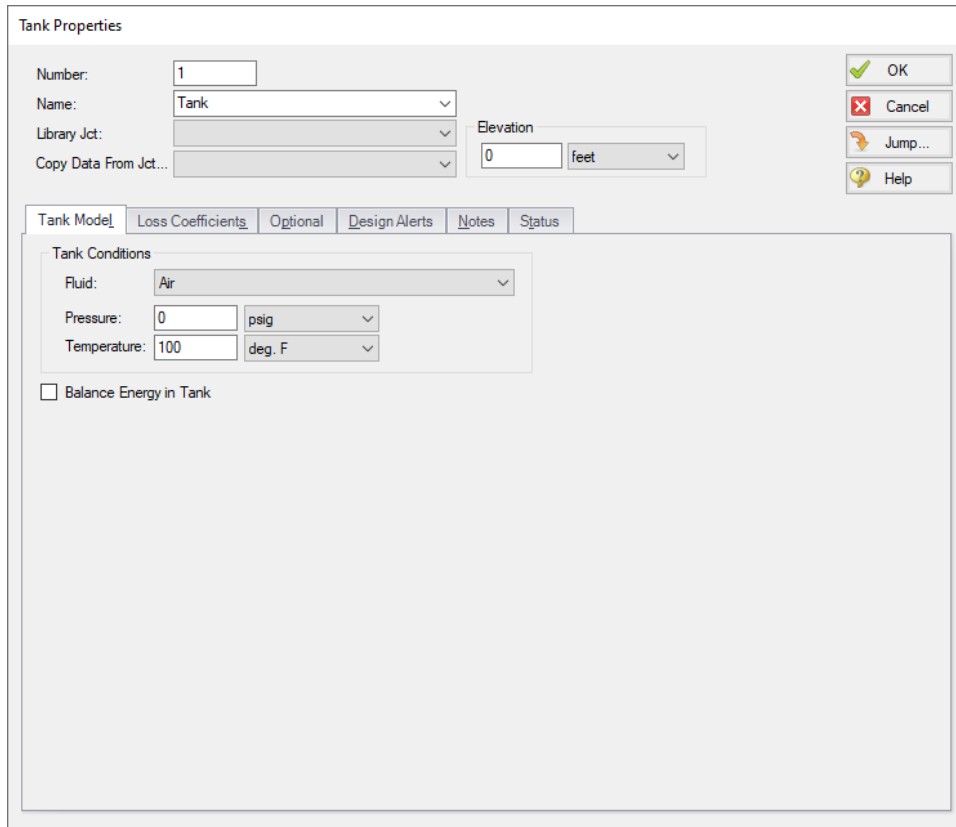
Click the Loss Coefficients tab to show the pipe table. The pipe table allows you to specify entrance and exit loss factors for each pipe connected to the tank (in this case there is only one). The default selection is the loss factors specified as zero. To later change the loss factors, click within the pipe table and enter the loss.

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 and temperature as well as other junction-specific data.

Each junction has a folder tab for notes, allowing you to enter text describing the junction or documenting any assumptions.

The Highlight feature displays all the required information in the Properties window in blue. The highlight is on by default. You can toggle the highlight off and on by pressing the F2 key. The highlight feature can also be turned on or off by selecting it on the User Options window. 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.



The screenshot shows the 'Tank Properties' dialog box. The 'Number' field is set to '1', 'Name' is 'Tank', 'Library Jct.' is empty, and 'Copy Data From Jct...' is empty. The 'Elevation' is set to '0' feet. The 'Tank Conditions' section shows 'Fluid' as 'Air', 'Pressure' as '0' psig, and 'Temperature' as '100' deg. F. There is an unchecked checkbox for 'Balance Energy in Tank'. The dialog has 'OK', 'Cancel', 'Jump...', and 'Help' buttons on the right. Below the main fields are tabs for 'Tank Model', 'Loss Coefficients', 'Optional', 'Design Alerts', 'Notes', and 'Status'.

**Figure 11: Properties window for Tank 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 the J1 tank icon and hold down the right mouse button. An information box appears, as shown in Figure 12.

Inspecting is a faster way of examining the input in an object rather than opening the Properties window.

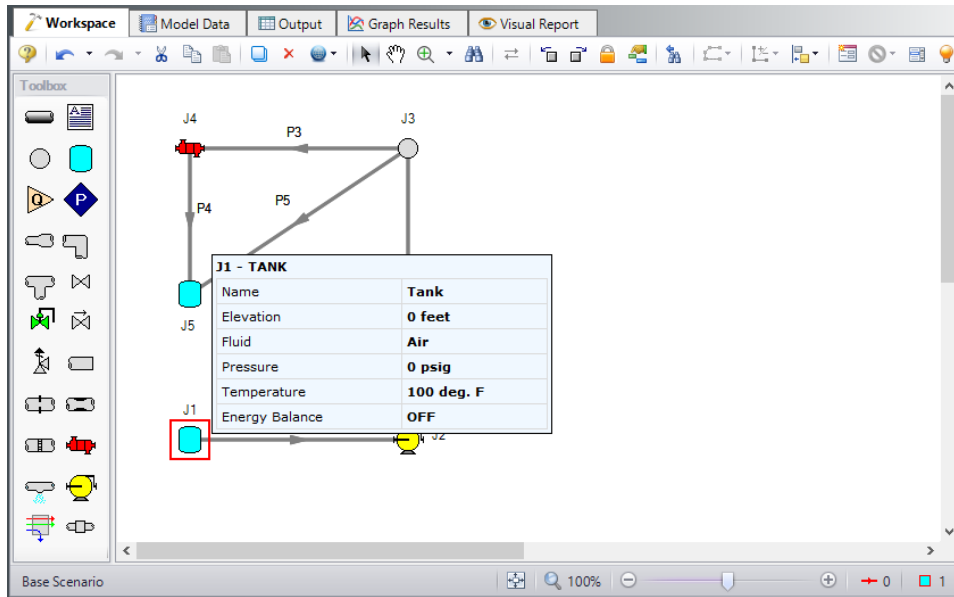


Figure 12: Inspecting from the Workspace with right mouse button

## VII. Define The Other Junctions

➤ **Next, open the properties window** for Compressor/Fan J2 (see Figure 13). Six compressor/fan models are provided. You can enter a compressor curve or you can enter a fixed mass flow rate or volumetric flow rate. A fixed flow rate will force the specified flow through the system and the resulting pressure rise will be calculated. A fixed pressure or head rise will apply the specified pressure or head at the compressor/fan and the resulting flow rate will then be calculated. A compressor/fan curve will model the actual performance of the compressor/fan. The pressure rise from a compressor/fan curve can be selected as either static or stagnation. The compression process models how the heat of compression affects the gas temperature, which is important in compressor applications.

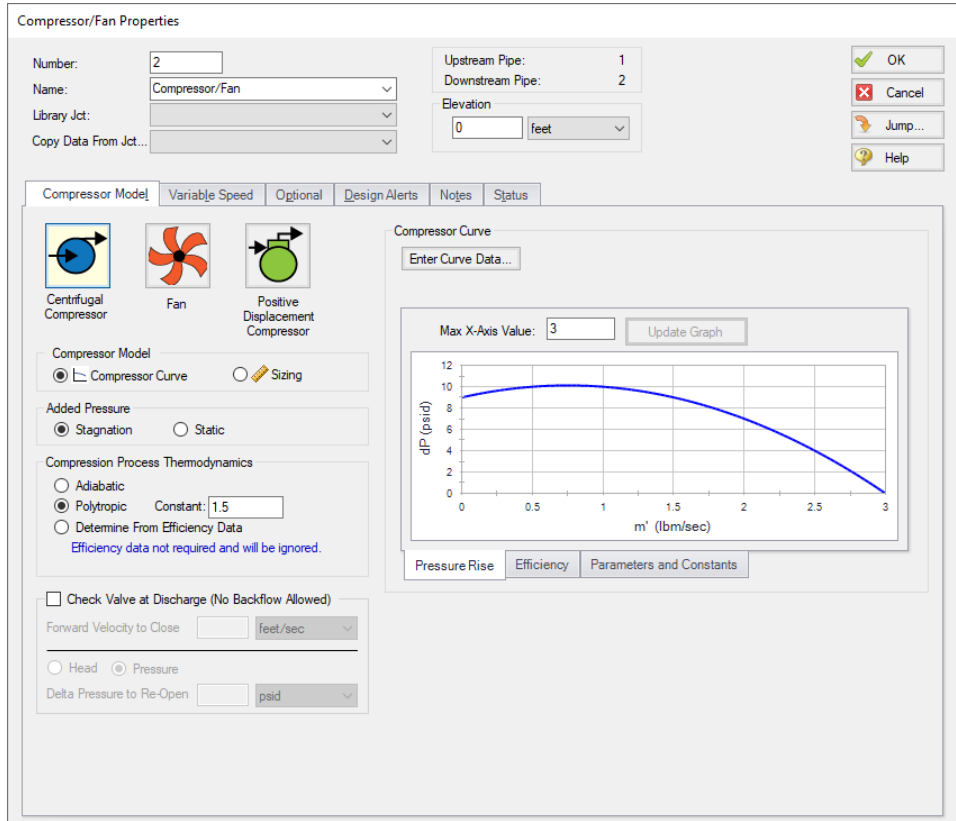
For this example, use Centrifugal Compressor and select Compressor Curve as the Compressor Model. Click Enter Curve Data and enter the following data into the Compressor Configuration window:

Mass	Pressure
lbm/sec	psid
0	9
1	10
2	7

Click Generate Curve Fit Now and then click OK. Make sure the Added Pressure selection is set to Stagnation.

## Walk-Through Examples

Set the Compression Process to Polytropic with a constant of 1.5. Last, enter an elevation of 0 feet. Then click OK and the compressor/fan should be completely defined. Figure 13 shows the Compressor/Fan Properties window.



**Figure 13: Properties window for Compressor/Fan J2**

- **Open the Branch J3 properties window** (see Figure 14). In this window, three connecting pipes should be displayed in the pipe table area. You can associate loss factors with each pipe by clicking within the pipe table and entering the data. These will be set to zero for this example.
- **Enter an elevation of 0 feet** for the Branch J3 (an elevation must be defined for all junctions). You could also click the Optional tab and then specify a flow source or sink at the junction. For this example, we will not use the Flow Source or Sink option in the Optional tab. Click OK and the branch is now defined.

Branch Properties

Number:

Name:

Library Jct:

Copy Data From Jct...

Elevation:

OK Cancel Jump... Help

Loss Coefficients Optional Design Alerts Notes Status

Pipe	Dir.	K - Flow into pipe	K - Flow out of pipe
2	In	0	0
3	Out	0	0
5	Out	0	0

Make All Pipes Same K Factor

Figure 14: Properties window for Branch J3

➤ **Open the Heat Exchanger J4 Properties window.** The thermal behavior of the heat exchanger is specified under the Thermal Data tab.

If you know the heat load for the exchanger, you can choose the Specified Heat Rate In thermal model and enter the heat rate directly. Adding heat to the system fluid is positive (i.e., heat flows out of the junction) while cooling is negative (heat flows into the junction).

You also have the option of choosing among a number of common heat exchanger configurations such as parallel and counter-flow. When you choose one of the other models, you must enter the secondary fluid conditions, heat exchanger area and heat transfer coefficient data. The secondary fluid can be either liquid or gas, although the models used to calculate heat transfer will be more accurate for liquids.

Whichever model is selected, the heat is added to or removed from the heat exchanger in accordance with the input model. The pressure drop through the heat exchanger is based on the Loss Model that is selected.

For this example, choose the Counter-Flow heat exchanger model with the following characteristics. Heat Transfer Area = 10 feet<sup>2</sup>, Overall Heat Transfer Coefficient = 10 Btu/hr-ft<sup>2</sup>-R, Secondary Fluid Flow Rate = 0.5 lbm/sec, Secondary Fluid Specific Heat = 1 Btu/lbm-R, and Secondary Fluid Inlet Temperature = 120 deg F.

The Thermal Data tab should now look like see Figure 15.

➤ **Go to the Loss Model tab and specify the K factor** at a value of 2. Enter an elevation of 0 feet, then click OK and the heat exchanger should be completely defined.

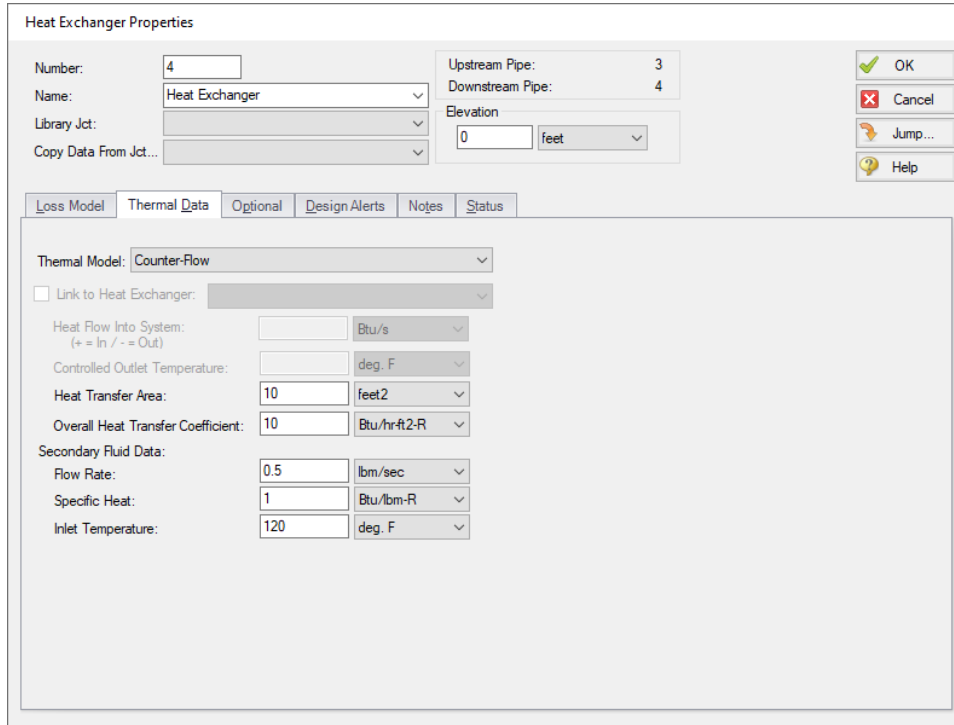


Figure 15: Properties window for Heat Exchanger J4

- **Open Tank J5** and enter a Pressure of 5 psig, a Temperature of 100 deg. F and an Elevation of 0 feet.
- **Save the model** again before proceeding.

### VIII. Define Pipe P1

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 choose Steel - ANSI from the Pipe Material drop down menu, 4 inch for Size, STD (schedule 40) for the Type, and a length of 10 feet.
- **On the Heat Transfer tab, select the Convective Heat Transfer model** and for the Ambient Conditions, set the Temperature to 40 deg. F the Fluid Velocity to 15 miles/hr, and the Fluid to Air (see Figure 17). Click OK. Pipe P1 is now defined.

## Walk-Through Examples

Pipe Properties

Number:  Upstream Junction: 1  
 Name:  Downstream Junction: 2  
 Copy Data From Pipe...:  Copy Previous...

OK  
 Cancel  
 Jump...  
 Help

Pipe Model | Fittings & Losses | **Heat Transfer** | Design Alerts | Optional | Notes | Status

Size

Pipe Material:  Length:    
 Pipe Geometry:   Model As Junction Connector  
 Size:   
 Type:   
 Inner Diameter:  inches  
 Outer Pipe ID:  inches  
 Inner Pipe OD:  inches  
 ID Reduction (Scaling):  % (optional)

Friction Model

Data Set:  User Specified  Standard  
 Absolute Roughness:  inches  
 Load Default

**Figure 16: Properties window for Pipe P1**

Pipe Properties

Number:  Upstream Junction: 1  
 Name:  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:  Ambient Conditions  
 Resistance Geometry:  Temperature:    
 Number of Insulation Layers:  Fluid Velocity:    
 Fluid:  Air  Water

Load Default

Resistance Type	Apply	Conductivity/Convection Data Source	Conductivity	Thickness	HT Area Ratio	Convection Coefficient
Units			Btu/hr-ft-R	inches		Btu/hr-ft <sup>2</sup> -R
Fluid Internal	<input checked="" type="checkbox"/>	Correlation			1	Gnielinski
Pipe Wall	<input checked="" type="checkbox"/>	Material Library	From Library	0.237	1	
External	<input checked="" type="checkbox"/>	Correlation			1	Forced (Churchill-Bernstein)

**Figure 17: Heat transfer data for Pipe P1**

### **The Pipe Properties window**

The Pipe Properties window offers control over all important flow and heat transfer parameters that are related to pipes.

The Inspection 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 the 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 Jump to jump to any other part of your model.

### IX. Define Other Pipes

➤ **Open the Pipe Properties window** for pipe P2 and choose pipe P1 from the Copy Data From Pipe drop-down menu (alternatively you can click the Copy Previous button). Choose all the properties and click OK. This will copy all pipe P1 input data to pipe P2. Then change the length of pipe P2 to 30 feet.

Open the other three Pipe Property windows and similarly copy all input data from pipe P1. Set the length of pipe P4 to 5 feet. Note that pipes P1, P3, and P5 will all have the same exact input data.

After completing the last pipe, all groups in Analysis Setup should be complete.

### X. Check Pipes And Junctions

➤ **Check if all the pipes and junctions are defined.** If not, turn on the Show Object Status from the View menu and open each undefined pipe and junction. The Status tab will indicate what information is missing.

### XI. Check Model Data

➤ **Before running the model, save it to file one more time.** It is also a good idea to review the input using the Model Data window.

#### *Reviewing input in the Model Data window*

The Model Data window is shown in Figure 18. To change to this window, you can select it from the Primary Window tabs or the Window menu, or by pressing CTRL+M. The Model Data window provides a tabular format for your model's input property data. Data in the Model Data window can be selected and then copied and pasted 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 area is called the General Section, the middle area is the Pipe Section, and the bottom area is the Junction Section.

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.



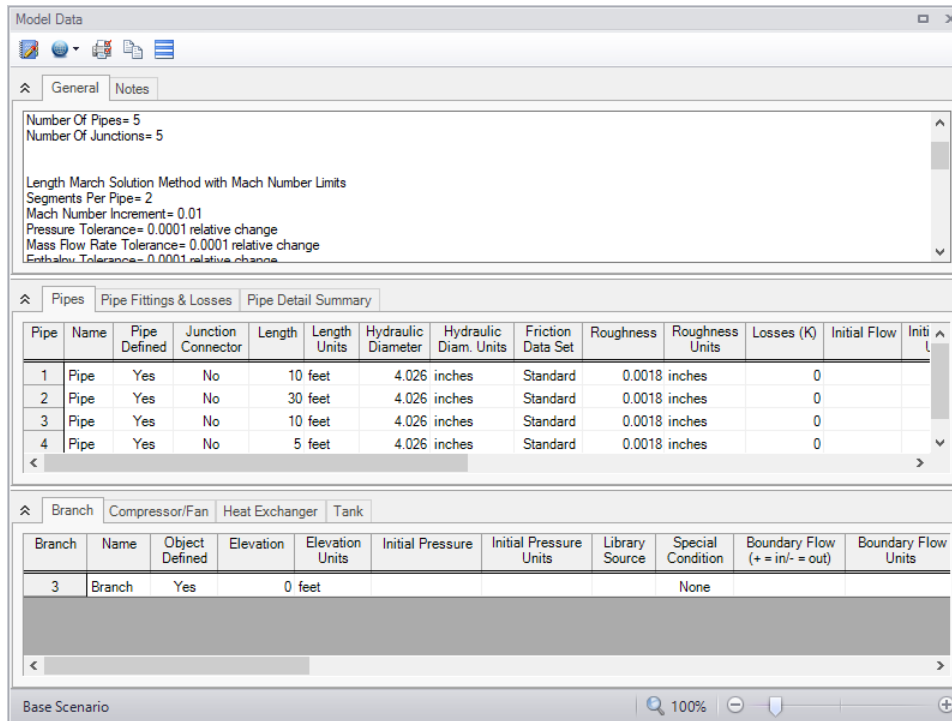


Figure 18: The Model Data window shows all input in text form

#### 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, Cost Settings, and Miscellaneous. All seven groups should have a green checkmark next to its name. Click OK.

#### Step 3. Run The Solver

➤ **Choose Run Model from the Toolbar** or Analysis menu. During execution, the Solution Progress window displays (see Figure 19). 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 20). 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: 2.94   
Solution Method: Length March with Mach # Limit - Length Increment = 2, Mach Limit = 0.01  
Relaxation: Automatic

	Absolute Tolerance Max Out of Tol.	Relative Tolerance Max Out of Tol.	Total Iterations
Pressure: 1.0E-04 Relative Change Not used (Absolute Change psia)	5.703E-08	2.870E-09	220
Mass Flow Rate: 1.0E-04 Relative Change Not used (Absolute Change lbm/sec)	6.090E-05	9.750E-05	67
Enthalpy: 1.0E-04 Relative Change Not used (Absolute Change Btu/lbm)	2.156E-07	9.911E-10	67
Concentration: 1.0E-04 Relative Change Not Used (Absolute Change)			

Determining connectivity...  
Initializing model...  
Checking if sonic choking exists based on initial guesses...  
Running Solver...  
Writing output to file...  
Calculations complete.

Other Actions ▼ Pause Cancel Graph Results... Visual Report... Output...

**Figure 19: The Solution Progress window displays the convergence progress**

## Walk-Through Examples

The Output window displays simulation results in text form and two data tables. The text output includes:

```

Execution Time= 2.71 seconds
Total Number Of Pressure Iterations= 220
Total Number Of Flow Iterations= 67
Total Number Of Enthalpy Iterations= 67
Number Of Pipes= 5
Number Of Junctions= 5
Matrix Method= Gaussian Elimination

Length March Solution Method with Mach Number Limits
Segments Per Pipe= 2
Mach Number Increment= 0.01
Pressure Tolerance= 0.0001 relative change
Mass Flow Rate Tolerance= 0.0001 relative change
    
```

The **Pipes** table shows the following data:

Pipe	Name	Mass Flow Rate (lbm/sec)	Vel. In (feet/sec)	Vel. Out (feet/sec)	Mach # In	Mach # Out	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. Inlet (deg. F)	T Stag. Inlet (deg. F)
1	Pipe	2.0551	342.57	353.82	0.29804	0.30854	0.47189	14.70	14.22	13.82	13.32	1.930	100.0
2	Pipe	2.0551	267.43	276.97	0.21770	0.22777	1.07264	20.94	19.87	20.26	19.17	12.382	175.7
3	Pipe	0.6246	82.22	81.00	0.06730	0.06685	0.03393	19.87	19.84	19.81	19.77	10.223	163.3
4	Pipe	0.6246	80.78	80.24	0.06696	0.06676	0.01675	19.71	19.70	19.65	19.63	4.568	147.6
5	Pipe	1.4314	190.30	190.36	0.15607	0.15679	0.17396	19.87	19.70	19.54	19.36	5.306	163.3

The **Junctions** table shows the following data:

Jct	Name	Mass Flow Rate Thru Jct (lbm/sec)	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. Inlet (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)
1	Tank	2.0551	0.0000	14.70	14.70	14.70	14.70	0.000	100.00	100.0	100.00	100.0
2	Compressor/Fan	2.0551	-6.7184	14.22	20.94	13.32	20.26	-77.588	98.07	175.7	87.63	169.7
3	Branch	N/A	0.0000	19.87	19.87	19.43	19.43	0.000	163.28	163.3	159.36	159.4
4	Heat Exchanger	0.6246	0.1235	19.84	19.71	19.77	19.65	5.443	153.06	147.6	152.51	147.1
5	Tank	N/A	0.0000	19.70	19.70	N/A	N/A	0.000	100.00	100.0	100.00	100.0

Figure 20: The Output window displays output in text form

### Step 4. Examine 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.

#### A. Specify Output Control

By default, AFT Arrow 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 Arrow (see Figure 1). A default title is also assigned in the Output Control.

➤ **Open Output Control** from the from the Common Toolbar or Tools menu. (Figure 21 shows the Output Control window). Click the General tab, enter a new title (if you like you can title this Air Heating System), 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 from the Display Parameters tab. You also can choose the units for the output.

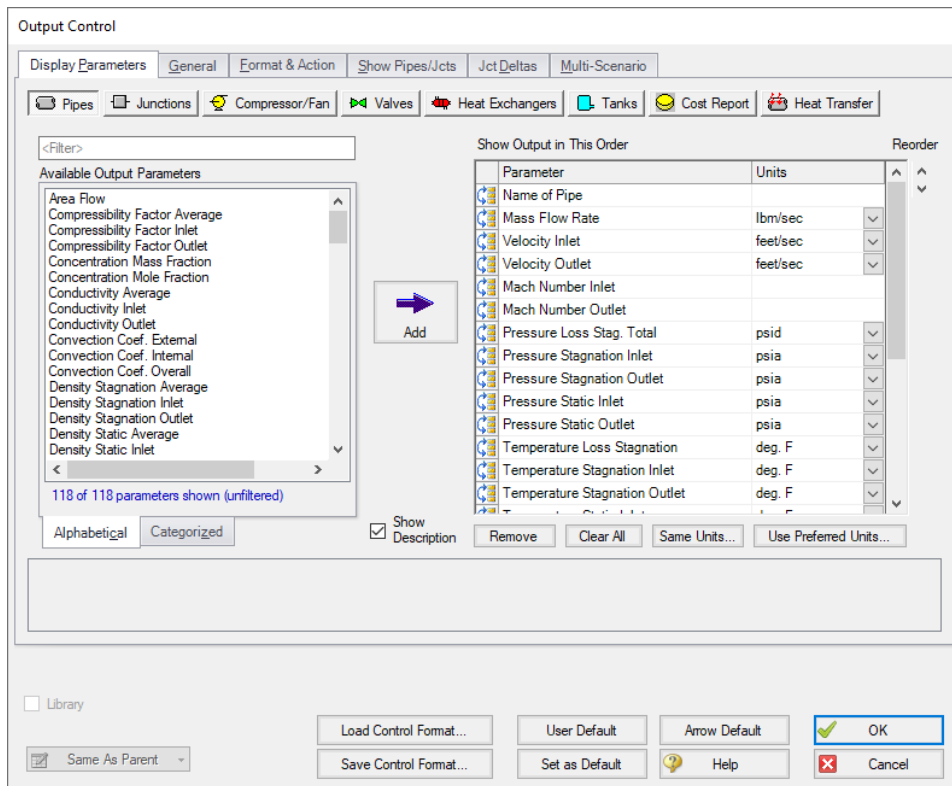


Figure 21: The Display Parameters tab allows you to customize which output data is displayed in AFT Arrow

## B. Modify The Output Format

If you selected the default AFT Arrow Output Control, the Pipe Results table will show mass flow rate in the second column with units of lbm/sec.

- **Select Output Control from the Tools menu.** On the right side of the Display Parameters tab for Pipes is the list of selected output parameters that will be displayed in the Output window. The left side shows parameters that are not in the current output report, but can be added at your option. You can view the list on the left either alphabetically or by categories by clicking the appropriate tab. On the list on the right, click Mass Flow Rate and change the units by selecting lbm/hr from the drop-down menu.
- **Click OK to display changes to the current results.** You should see the mass flow rate results, still in the second column, in units of lbm/hr. Notice the Velocity Inlet (Vel. In) results in the third column.
- **Select Output Control from the Tools menu** one more time. Select the Pipes in the Display Parameters tab. The parameters can be reordered by clicking the Reorder arrows on the right of the list, or by dragging the Reorder parameter icon on the left of the Output parameter you would like to reorder.
- **Select the Velocity Inlet parameter** and 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 Vel. In (Velocity Inlet), the second column now contains the pipe name, and the third column the mass 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 Arrow in the way that is most meaningful to you, reducing the possibility of errors.

➤ **Lastly, double-click the column header for Vel. In in the Output window Pipe Results Table.**

This will open a window in which you can change the units once again if you prefer. These changes are extended to the Output Control parameter data you have previously set.

### C. View The Visual Report

➤ **Change to the Visual Report window** by clicking the Visual Report tab or by choosing it from the Windows menu. This window allows you to integrate your text results with the graphic layout of your pipe network.

➤ **Click the Visual Report Control button on the Toolbar** (or Tools menu) and open the Visual Report Control window, shown in Figure 22. The default list of parameters are filtered such that only the parameters currently displayed with the Output Control are shown. You can also set the Parameter Filter such that All Available Parameters will be visible. For now, select Velocity Outlet in the Pipe Results list and Pressure Static Inlet in the Junction Results list. Click the Show button. The Visual Report window graphic is generated as shown in Figure 23.

It is common for the text in the Visual Report window to overlap when first generated. You can change this by selecting smaller fonts or by dragging the text to a new area to increase clarity (this has already been done in Figure 23). This window can be printed or copied to the clipboard for import into other Windows graphics programs, saved to a file, or printed to an Adobe PDF file.

## Walk-Through Examples

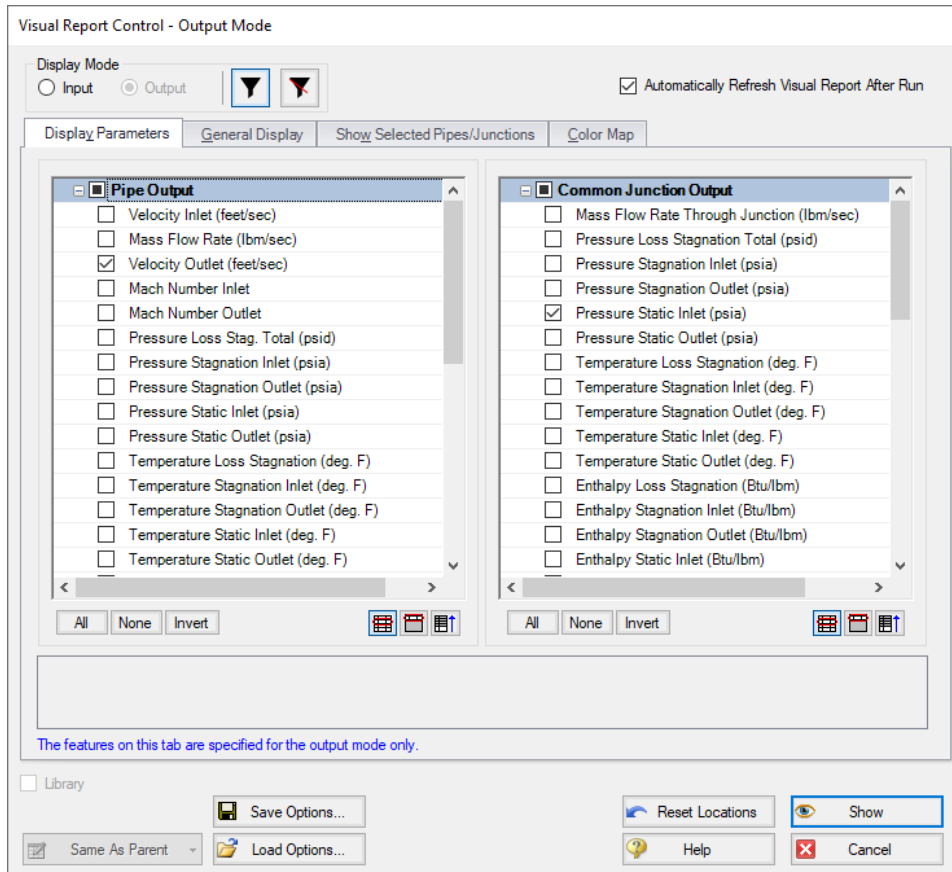


Figure 22: The Visual Report Control window selects content for the Visual Report window

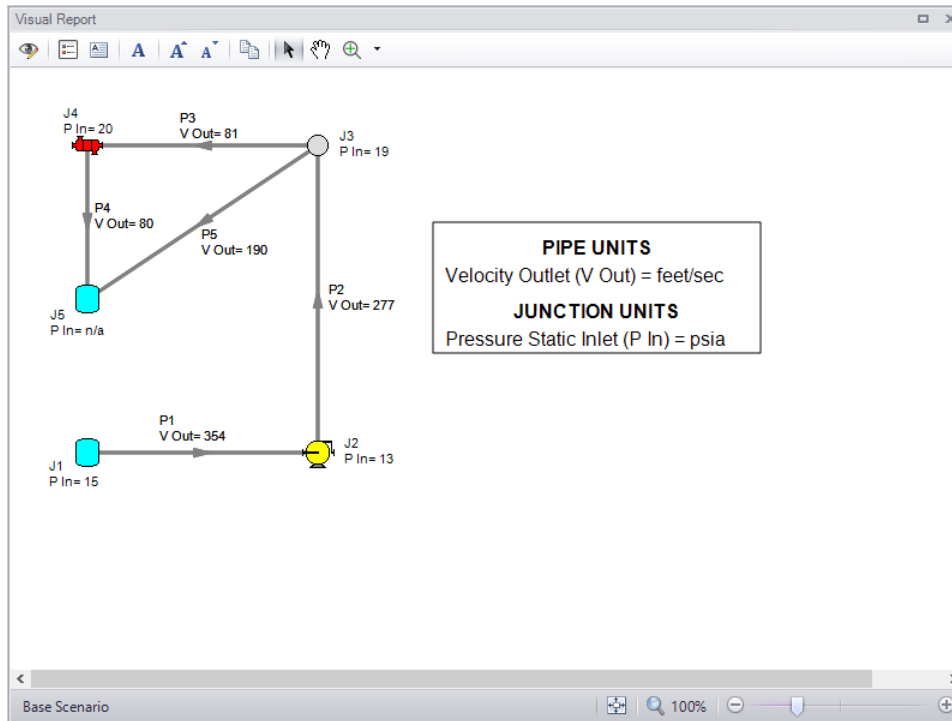


Figure 23: The Visual Report window integrates results with model Layout

## D. Graph The Results

➤ **Change to the Graph Results window** by clicking the Graph Results tab, choosing it from the Windows menu, or by pressing CTRL+G. The Graph Results window offers full-featured Windows plot preparation.

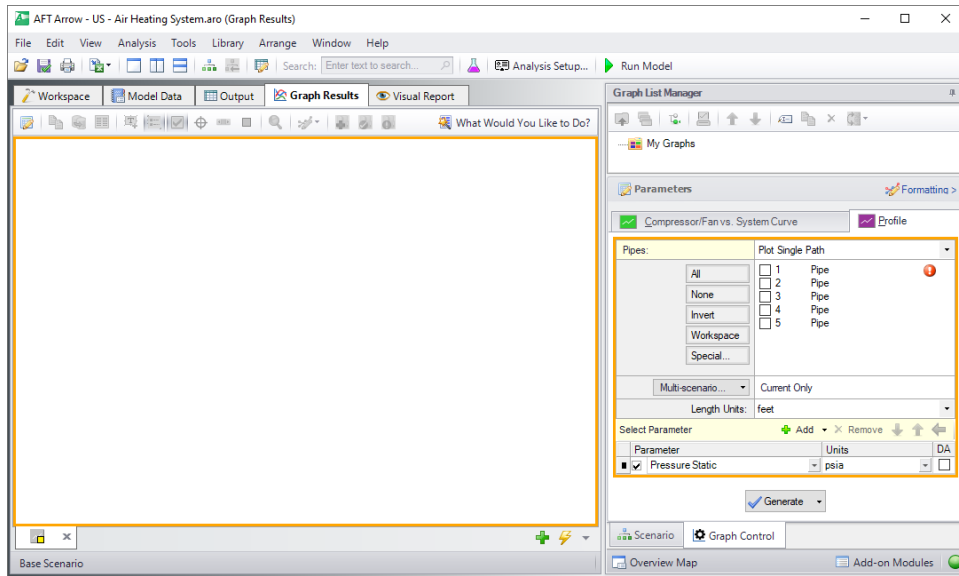
The Graph Parameters section will automatically be displayed in the Graph Control tab on the Quick Access Panel on the far right of the Graph Results window. The Graph Parameters section on the Graph Control tab is where you can specify which graphs you would like to generate. Multiple types of graphs can be generated such as Profiles, Compressor/Fan vs. System Curves, System Curves, or Solutions.

Select the Profile Graph tab in the Graph Parameters area in the Graph Control on the Quick Access Panel. Plot Single Path selected by default, select pipes P1 through P4 from the Pipes list. Choose Pressure Static, if not already selected, from the Parameter drop-down menu. Make sure feet is specified as the Length Units and psia as the Pressure Static Units. Then click Generate. The Show Junction Locations feature is located on the Graph Results Toolbar and can be enabled or disabled by clicking the icon (see Figure 24).

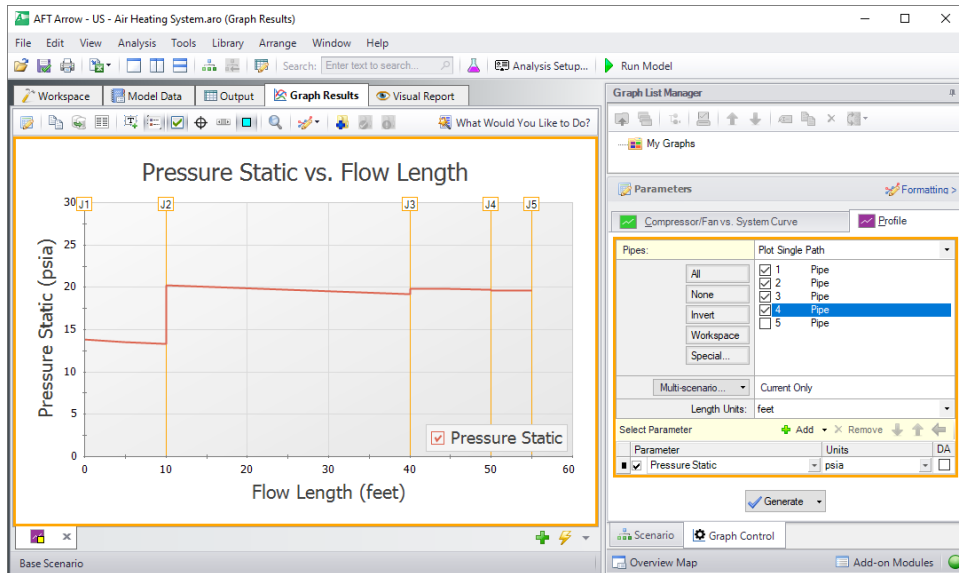
A graph appears showing the static pressure along the flow-path defined by pipes P1 through P4 in Figure 25.

You can use the other buttons in the Graph Results window to change the graph appearance and to save and export 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.

## Walk-Through Examples



**Figure 24: The Graph Parameters section of the Graph Control tab on the Quick Access Panel controls the Graph Results content**



**Figure 25: The Graph Results window offers full-featured plot generation**

You have now used AFT Arrow's five Primary Windows to build a simple model. Review the rest of the Arrow Help File contents for more detailed information on each of the windows and functions.



# Flow Through an Orifice

## Summary

---

This example demonstrates a sample calculation to determine the maximum flow through a system where sonic choking occurs, as well as the sonic pressure and area, and it explores the effect of CdA on the system flow rate.

## Topics Covered

- Determining the maximum flow through a system
- Determining sonic (stagnation) pressure
- Determining sonic area
- Using scenarios

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

## Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Arrow installation:

- ***US - Orifice Flow.aro***

## Problem Statement

For this problem, steam flows from one tank to another, through an orifice.

The first pipe from the inlet tank to the orifice is 10 feet long, and the second pipe leading from the orifice to the discharge tank is 20 feet long. Both pipes are 2 inch, STD (schedule 40) pipes, and are well insulated (adiabatic).

The inlet tank has a pressure of 250 psig and a temperature of 500 deg. F. The discharge tank has a pressure of 0 psig. Set the discharge temperature to 500 deg. F. The discharge temperature is not important in this case because no fluid is entering the system at the discharge, so the fluid temperature cannot impact the system temperature. Both tanks are at 0 feet elevation.

The orifice at the end of the first pipe has an area of 3 inches<sup>2</sup>, and a K Factor of 10.

Determine the following:

1. What is the maximum flow through the system?
2. What is the sonic (stagnation) pressure at the system exit?
3. At the maximum flow, what is the sonic area at the orifice? At the discharge tank?

### **Step 1. Start AFT Arrow**

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

### **Step 2. Define the Fluid Properties Group**

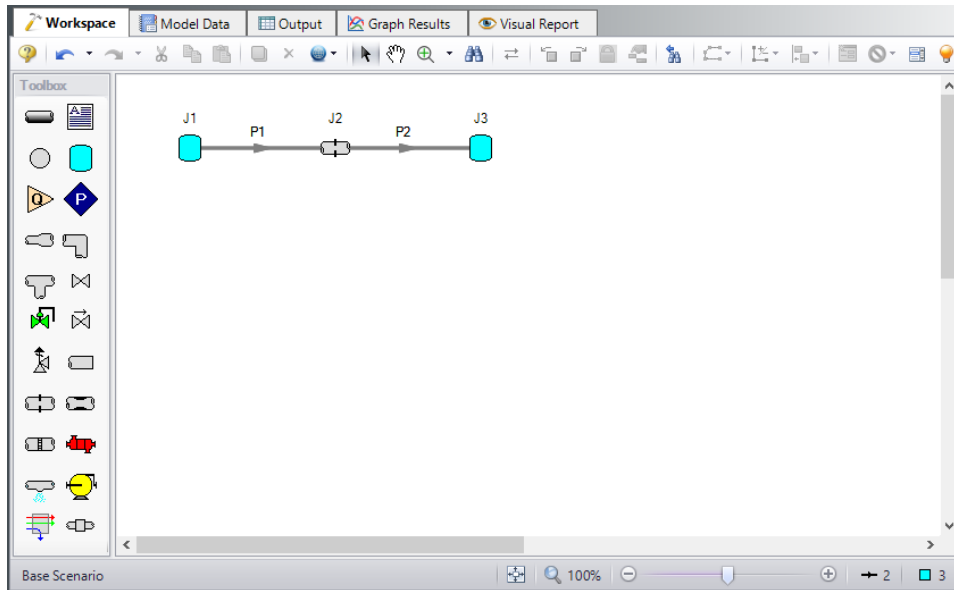
---

1. Navigate to the Fluid panel in Analysis Setup
2. Define the Fluid panel with the following inputs
  - a. Fluid Library = AFT Standard
  - b. Fluid = Steam
    - i. After selecting, click Add to Model
  - c. Equation of State = Redlich-Kwong
  - d. Enthalpy Model = Generalized
  - e. Specific Heat Ratio Source = Library

### **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 window showing the model layout for Flow Through an Orifice 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.

## 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. Lengths =

Pipe	Length (feet)
1	10
2	20

## Junction Properties

1. J1 Tank
  - a. Name = Inlet Tank
  - b. Elevation = 0 feet

- c. Fluid = Steam
  - d. Pressure = 250 psig
  - e. Temperature = 500 deg. F
2. J2 Orifice
- a. Elevation = 0 feet
  - b. Orifice Type = User Specified
  - c. Subsonic Loss Model = K Factor
  - d. Orifice Dimensions = Area
  - e. Area = 3 inches<sup>2</sup>
  - f. K = 10
3. J3 Tank
- a. Name = Discharge Tank
  - b. Elevation = 0 feet
  - c. Fluid = Steam
  - d. Pressure = 0 psig
  - e. Temperature = 500 deg. F

➤ **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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

### Step 5. Specify the Output Control

---

Open the Output Control window by selecting Output Control from the Toolbar or Tools menu.

The Output window (Figure 2) contains all the data that is specified in the Output Control window. The window starts with a default set of outputs, which is normally sufficient. You can also specify the units for each output parameter from this window. For this example, change the Pressure Stagnation Inlet units to psig (after making the change, click Same Units below the Output list, then click OK to change all pressure units to psig. Next, click the Junctions button at the top and change the Sonic Flow Area units to inches<sup>2</sup>.

### Step 6. Examine the Output

---

Because this system chokes at the discharge tank, the Sonic Choking tab appears in the General section of the Output window. This tab will appear when sonic choking occurs in the system, and will contain

# Flow Through an Orifice

messages pertaining to the sonic choking conditions.

Because the flow is choked at the exit, the mass flow rate shown in the Output reflects the maximum flow rate through the system for the given conditions. The maximum flow rate through this system is 4.646 lbm/sec. The stagnation pressure at the exit is 88.27 psia. The output in the junction data portion of the Output window shows the Sonic Area at the orifice is 1.377 inches<sup>2</sup>, and the Sonic Area at the discharge tank is 3.355 inches<sup>2</sup>.

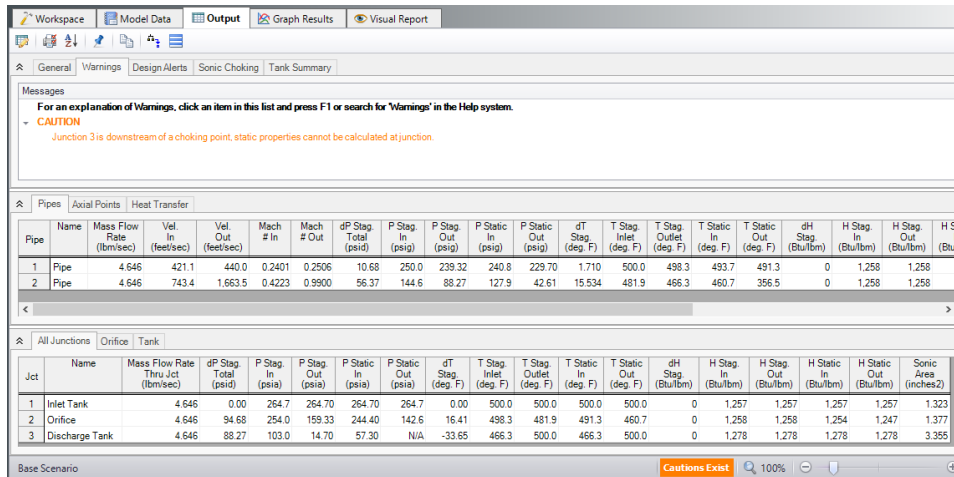


Figure 2: Output window showing results for Flow Through an Orifice Example

## Step 7. System Modification

Set the CdA (for Sonic Choking - Optional) in the Orifice Properties window to 20% higher than the sonic area and then 20% lower than the sonic area, and rerun the model for each scenario. How do the results change for each case?

**Note:** This is a situation where a user can create two new scenarios using the Scenario Manager to examine several what-if situations, without disturbing the basic model.

## Step 8. Re-evaluate the Model

Setting the orifice CdA higher than the sonic area has no effect on the model because the system is already choking at the exit. Setting the CdA 20% lower than the sonic area (1.102 inches<sup>2</sup>) causes the flow rate to decrease to 3.770 lbm/sec, and the exit discharge stagnation pressure to decrease to 68.77 psig. Also, lowering the CdA has caused the system to choke in two places rather than just one. This can be seen on the Sonic Choking tab in the General section of the Output window.

# Control Valve

## Summary

---

This example will walk you through a simple calculation to calculate a pipe size for a system with a control valve.

## Topics Covered

- Using flow control valves
- Entering and changing pipe size data
- Adding items to the output using Output Control
- Modifying output using Output Control

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

## Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Arrow installation:

- ***US - Control Valve.aro***

## Problem Statement

For this problem, steam flows from one tank to another. The pipe between the tanks is horizontal, and 450 feet long. A control valve is used to control the flow. The control valve must have a pressure drop of at least 20 psid to operate correctly. Assume the control valve is in the middle of the pipe. The ambient temperature is 60 deg. F. The pipe has an ambient air velocity of 15 miles/hr, and no insulation. The inlet stagnation pressure and temperature values are 240 psig and 425 deg. F, and the outlet stagnation pressure and temperature values are 140 psig and 425 deg. F. Note that the discharge temperature does not have an impact on calculations unless the discharge boundary happens to be supplying flow. Therefore, an assumed temperature can be used.

The pipe must be sized correctly to allow 5 lbm/sec to flow from one reservoir to the other. Consider only STD (schedule 40) pipes.

---

## Step 1. Start AFT Arrow

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

---

## Step 2. Define the Fluid Properties Group

---

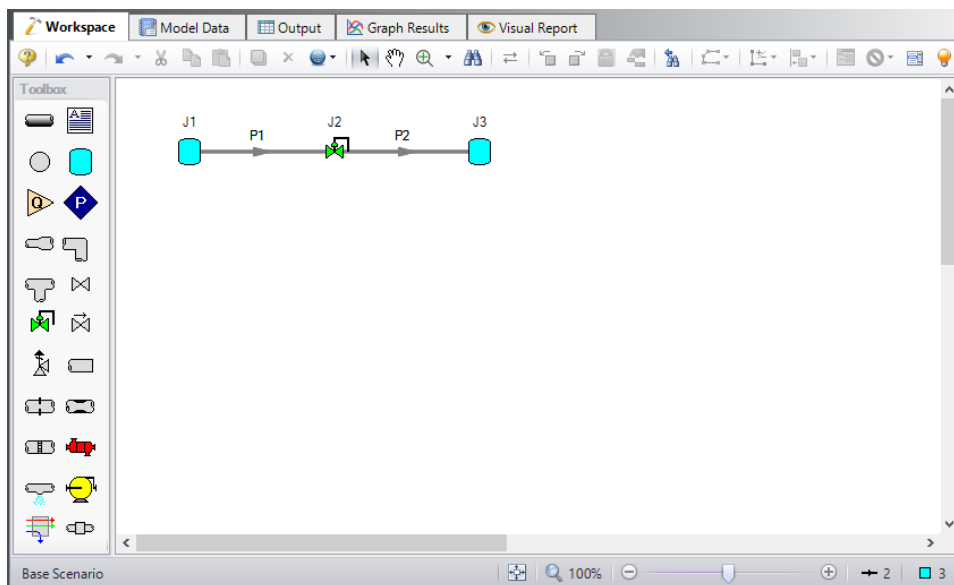
1. Navigate to the Fluid panel in Analysis Setup
2. Define the Fluid panel with the following inputs
  - a. Fluid Library = AFT Standard
  - b. Fluid = Steam
    - i. After selecting, click Add to Model
  - c. Equation of State = Redlich-Kwong
  - d. Enthalpy Model = Generalized
  - e. Specific Heat Ratio Source = Library

---

## 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 window showing the model layout for Control Valve 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.

For this example, you need to find the correct pipe size but you are limited to STD (schedule 40) pipe sizes. You need to use your engineering judgment to take an initial guess at the pipe size for 5 lbm/sec. Try 2-½ inch pipe. Specify the pipes as follows:

## Pipe Properties

1. Pipe Model tab
  - a. Pipe Material = Steel - ANSI
  - b. Pipe Geometry = Cylindrical Pipe
  - c. Size = 2-1/2 inch
  - d. Type = STD (schedule 40)
  - e. Friction Model Data Set = Standard
  - f. Length = 225 feet
2. Heat Transfer tab
  - a. Heat Transfer Model = Convective Heat Transfer
  - b. Temperature = 60 deg. F
  - c. Fluid Velocity = 15 miles/hr
  - d. Fluid = Air

Resistance Type	Apply	Conductivity/Convection Data Source	Conductivity	Thickness	HT Area Ratio	Convection Coefficient
Units			Btu/hr-ft-R	inches		Btu/hr-ft <sup>2</sup> -R
Fluid Internal	<input checked="" type="checkbox"/>	Correlation			1	Gnielinski
Pipe Wall	<input checked="" type="checkbox"/>	Material Library	From Library	0.203	1	
External	<input checked="" type="checkbox"/>	Correlation			1	Forced (Churchill-Bernstein)

Figure 2: Pipe Heat Transfer Properties window

## Junction Properties



1. J1 Tank
  - a. Elevation = 0 feet
  - b. Fluid = Steam
  - c. Pressure = 240 psig
  - d. Temperature = 425 deg. F
2. J2 Control Valve
  - a. Elevation = 0 feet
  - b. Valve Type = Flow Control (FCV)
  - c. Control Setpoint = Mass Flow Rate
  - d. Flow Setpoint = 5 lbm/sec
  - e. Always Control (Never Fail) = Unchecked
3. J3 Tank
  - a. Elevation = 0 feet
  - b. Fluid = Steam
  - c. Pressure = 140 psig
  - d. Temperature = 425 deg. F

➤ **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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

### Step 5. Specify the Output Control

---

Open the Output Control window by selecting Output Control from the Toolbar or Tools menu.

In this example, you want to review the effects of changing the pipe nominal size. To simplify this, add this parameter to the output.

1. Click the Pipes button in the Display Parameters tab.
2. In the list on the left, find the parameter Nominal Size.
3. Select the Nominal Size parameter and then click the Add button, which adds the parameter to the list of displayed parameters on the right.
4. 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 3).
5. Click OK.

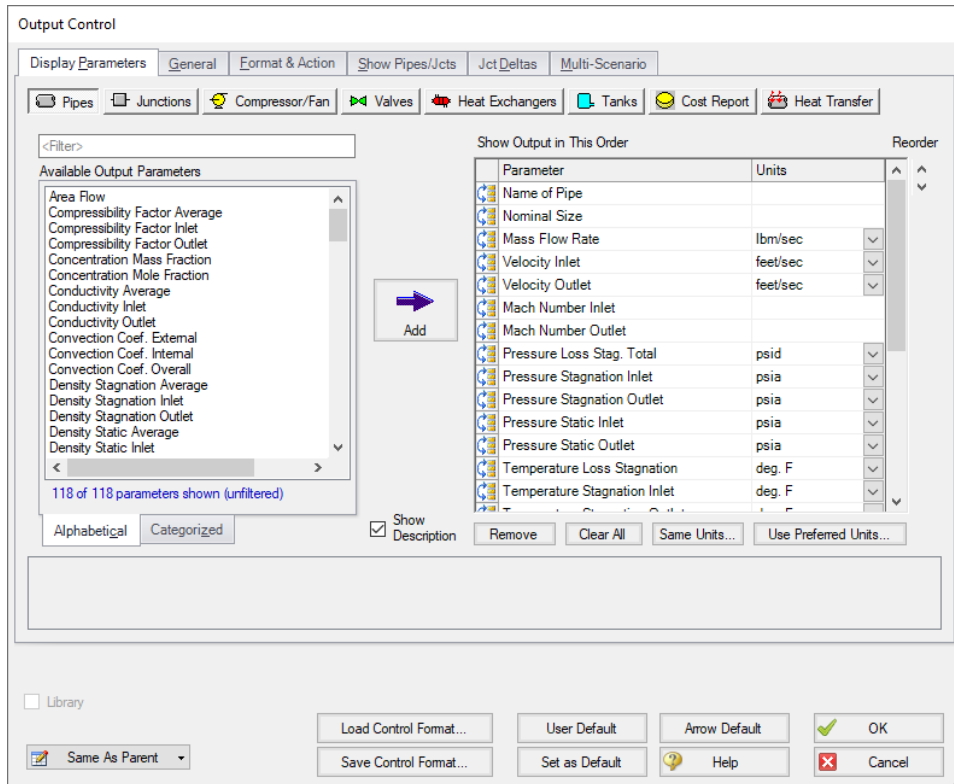


Figure 3: Output Control window

## Step 6. Examine the Output

The Output window contains all the data that was specified in the Output Control window. The output is shown in Figure 4. In the General section of the Output window, there is a Warnings tab. Select this tab and you can see that for this case, the flow control valve was not able to achieve the required flow rate of 5 lbm/sec. The pressure drop across the valve is higher than the pressure drop required for the system to balance, so the valve is unable to maintain control for this size of pipe.

In addition warnings are shown that the fluid is above the saturation pressure line. AFT Arrow cannot calculate the actual behavior of the gas when the saturation line is reached. Instead Arrow will extrapolate the fluid properties assuming that the fluid would remain in the gas phase, and will warn the user that the saturation line was exceeded as is seen here.

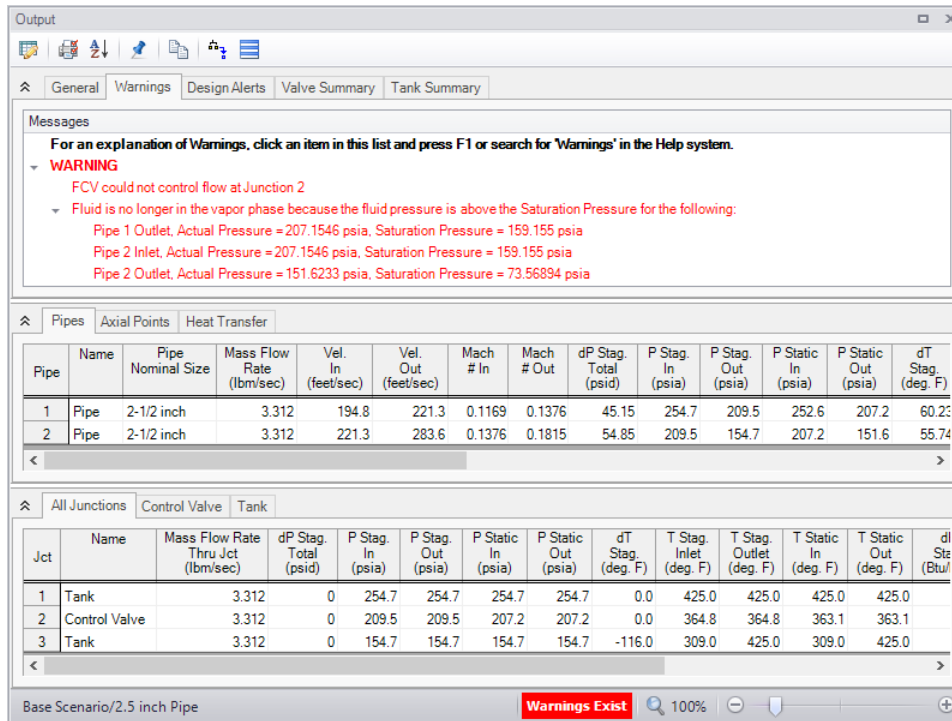


Figure 4: Output for Control Valve Example

## Step 7. Size Piping Insulation

It is necessary to evaluate a new pipe size and to address the condensation warnings. Let's first address the condensation warnings, the pipe size will be addressed in the next step. We will assume that the necessary insulation thickness will not vary largely with pipe size because the expected change in pipe size should be relatively small in the next step.

One way to keep the fluid temperature above saturation and prevent condensation, would be to add insulation to the pipe. Assume there are three sizes of insulation available, 0.25 inch, 0.5 inch, and 1.0 inch thick, each of which has a thermal conductivity of 0.1 Btu/hr-ft-R. Assuming the same insulation for both pipes, which insulation size would be necessary to prevent condensation in the line?

Because this 2-1/2 inch model is valuable to refer to, keep this scenario by returning to the Workspace and then creating a new scenario on the Scenario Manager as a Child scenario by right-clicking on the Base Scenario and selecting Create Child. Name the child scenario *With Insulation*. Insulation is added on the Pipe Properties window's Heat Transfer tab. AFT Arrow allows you to specify up to three layers of insulation. The insulation can be specified as internal insulation, external insulation, or a combination of the two. Add the insulation to the pipe by changing the Number of Insulation Layers to 1 External, and then add the insulation thermal data to the table (see Figure 5). For Conductivity, enter 0.1 Btu/hr-ft-R and for thickness, enter 0.5 inches as an initial guess. Make these changes to both pipes.

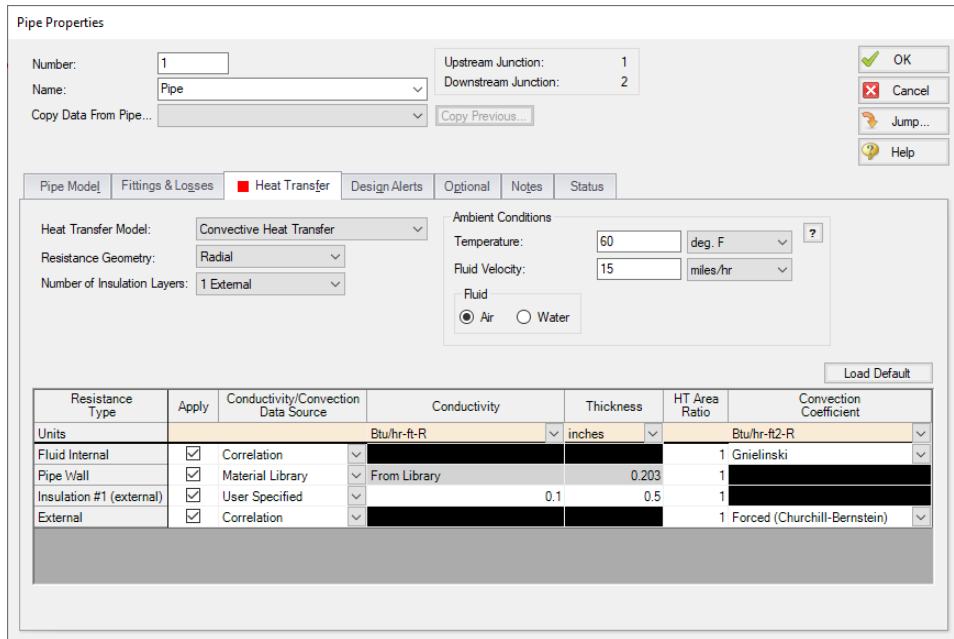


Figure 5: Insulation data is added on the Heat Transfer tab of the Pipe Properties window

## Step 8. Change Pipe Size

Create a child scenario of the *With Insulation* scenario and name it *3 inch Pipe*. From this scenario, select a new pipe size in the Pipe Properties window for both pipes. By trial and error, you can determine that by using 3-inch pipe, the minimum 20 psid pressure drop requirement can be satisfied (see Valve Summary tab in Figure 6). Do the results change if the valve is not in the middle of the pipe?

The screenshot shows the 'Valve Summary' tab with a table of output results for a 3-inch nominal pipe size modified control valve example.

Jct	Name	Valve Type	Mass Flow (lbm/sec)	dP Stag. (psid)	dP Static (psid)	P Static In (psia)	K	Cv	Cv Est.	Valve State
2	Control Valve	FCV	5.000	21.98	22.24	219.4	9.719	N/A	97.20	Open

Figure 6: Output results for 3 inch Nominal Pipe Size modified Control Valve Example

## Conclusion

To meet the system requirements, 3-inch pipe should be used. If the valve were not in the middle of the pipe, the results would change because there is less pressure drop available as the valve is moved towards the pipe inlet. To prevent condensation in the line, the 0.5-inch thick insulation would have to be used.

# Natural Gas Burner

## Summary

---

This example will walk you through an example to size a heat exchanger to deliver natural gas at a minimum required temperature.

## Topics Covered

- Sizing a Heat Exchanger
- Entering a loss curve for a Heat Exchanger
- Specifying heat transfer data for pipes

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

## Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Arrow installation:

- ***US - Natural Gas Burner.aro***

## Problem Statement

An underground storage reservoir containing natural gas made up mostly of methane supplies gas to five burners. The gas is at 500 psig, and 50 deg. F. The gas is supplied to each of the burners at 100 psig. To ensure that the flow rate to each of the burners is equal, each burner has a control valve that controls the flow to 5 lbm/sec. The pipes in the system are uninsulated STD (schedule 40) steel, with standard roughness. The ambient temperature surrounding the pipes is 75 deg. F, and the pipes have an ambient air velocity of 15 miles/hr.

In order to operate at the best efficiency, the gas should be delivered to the burners at a minimum stagnation temperature of 100 deg. F. In order to achieve this, the gas flows through a heat exchanger, which has a 10 psid pressure drop at 25 lbm/sec.

How much heat must be added to the heat exchanger to ensure that all of the burners will receive gas at the specified minimum temperature? (Neglect elevation changes)

### **Step 1. Start AFT Arrow**

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

### **Step 2. Define the Fluid Properties Group**

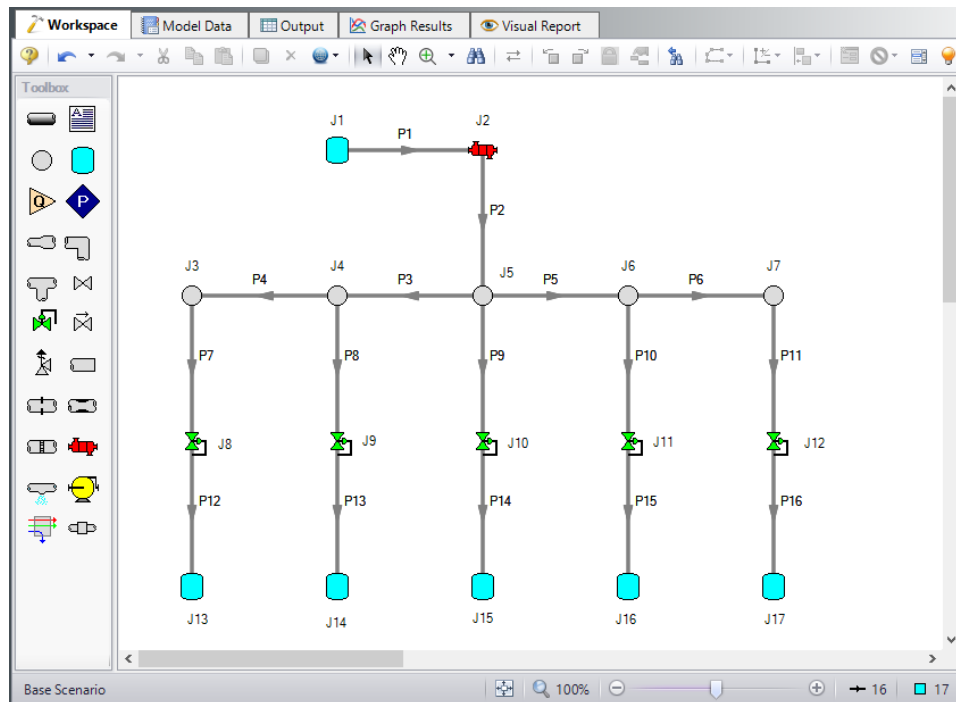
---

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 = Methane
    - i. After selecting, click Add to Model
  - c. Equation of State = Redlich-Kwong
  - d. Enthalpy Model = Generalized
  - e. Specific Heat Ratio Source = Library

### **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 Natural Gas Burner 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.

## 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 (feet)
1	4 inch	100
2-6	4 inch	20
7-16	3 inch	4

2. Heat Transfer tab
  - a. Heat Transfer Model = Convective Heat Transfer
  - b. Temperature = 75 deg. F
  - c. Fluid Velocity = 15 miles/hr
  - d. Fluid = Air

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: 75 deg. F  
 Fluid Velocity: 15 miles/hr  
 Fluid:  Air  Water

Load Default

Resistance Type	Apply	Conductivity/Convection Data Source	Conductivity	Thickness	HT Area Ratio	Convection Coefficient
Units			Btu/hr-ft-R	inches		Btu/hr-ft <sup>2</sup> -R
Fluid Internal	<input checked="" type="checkbox"/>	Correlation			1	Gnielinski
Pipe Wall	<input checked="" type="checkbox"/>	Material Library	From Library	0.237	1	
External	<input checked="" type="checkbox"/>	Correlation			1	Forced (Churchill-Bernstein)

Figure 2: Pipe Heat Transfer Data window

## Junction Properties

1. All Control Valves
  - a. Elevation = 0 feet
  - b. Valve Type = Flow Control (FCV)
  - c. Control Setpoint = Mass Flow Rate
  - d. Flow Set Point = 5 lbm/sec
2. J1 Tank
  - a. Elevation = 0 feet
  - b. Fluid = Methane
  - c. Pressure = 500 psig
  - d. Temperature = 50 deg. F
3. J13-J17 Tanks
  - a. Elevation = 0 feet
  - b. Fluid = Methane



- c. Pressure = 100 psig
- d. Temperature = 500 deg. F
- 4. J3-J7 Branches
  - a. Elevation = 0 feet
- 5. J2 Heat Exchanger
  - a. Elevation = 0 feet
  - b. Loss Model = Resistance Curve
  - c. Enter Curve Data =

<b>Mass</b>	<b>Pressure</b>
<b>lbm/sec</b>	<b>psid</b>
0	0
25	10
50	40

- d. Curve Fit Order = 2
- e. Click Generate Curve Fit Now

To solve this problem, you will have to guess how much heat to add to the heat exchanger, examine the results, and adjust the heat added until you achieve the desired minimum temperature at the burners. To do this, select the Heat Transfer tab on the Heat Exchanger Properties window, and for the Thermal Model, select Specified Heat Rate In Constant. Next, enter a value for Heat Flow Into System. For the first guess, use 500 Btu/s.

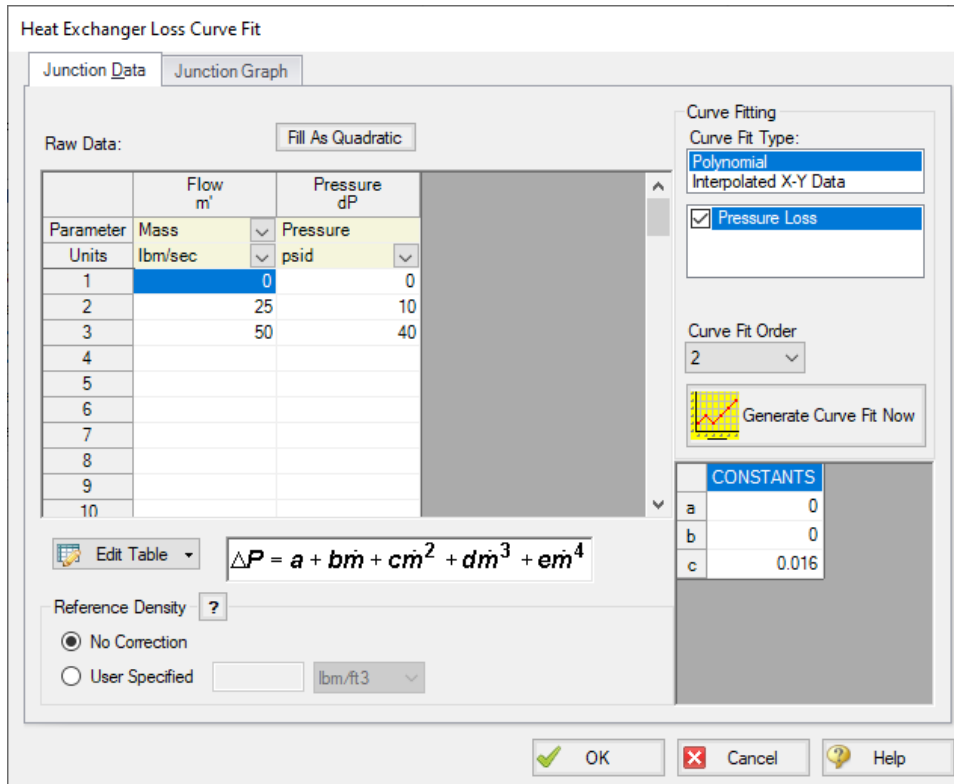


Figure 3: Heat Exchanger Loss Curve Fit

➤ **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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

#### Step 5. Examine the Output

Examination of the stagnation temperature at the outlets of the pipes discharging to the burners shows that the lowest temperature is 64.08 deg. F (see Figure 4). This is well below the required minimum temperature of 100 deg. F. The heat added to the system fluid from the heat exchanger must be increased, and the model ran again to determine the outlet temperature at the value.

# Natural Gas Burner

Pipe	Name	Mass Flow Rate (lbm/sec)	Vel. In (feet/sec)	Vel. Out (feet/sec)	Mach # In	Mach # Out	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)
1	Pipe	25.000	174.76	184.80	0.13091	0.13815	26.9921	514.7	487.7	509.3	482.0	1.20679	50.00	48.79
2	Pipe	25.000	205.22	207.98	0.14664	0.14858	6.2206	477.7	471.5	471.4	465.1	0.32922	83.08	82.75
3	Pipe	10.000	82.47	82.64	0.05885	0.05896	0.9867	471.5	470.5	470.5	469.5	0.09753	83.08	82.98
4	Pipe	5.000	41.27	41.28	0.02944	0.02945	0.2479	470.5	470.2	470.2	470.0	0.10858	83.03	82.92
5	Pipe	10.000	82.47	82.64	0.05885	0.05896	0.9867	471.5	470.5	470.5	469.5	0.09753	83.08	82.98
6	Pipe	5.000	41.27	41.28	0.02944	0.02945	0.2479	470.5	470.2	470.2	470.0	0.10858	83.03	82.92
7	Pipe	5.000	71.14	71.17	0.05077	0.05079	0.2044	470.2	470.0	469.5	469.3	0.02649	82.89	82.86
8	Pipe	5.000	71.12	71.15	0.05074	0.05077	0.2043	470.5	470.3	469.7	469.5	0.02667	83.00	82.97
9	Pipe	5.000	70.98	71.01	0.05064	0.05066	0.2039	471.5	471.3	470.7	470.5	0.02692	83.10	83.07
10	Pipe	5.000	71.12	71.15	0.05074	0.05077	0.2043	470.5	470.3	469.7	469.5	0.02667	83.00	82.97
11	Pipe	5.000	71.14	71.17	0.05077	0.05079	0.2044	470.2	470.0	469.5	469.3	0.02649	82.89	82.86
12	Pipe	5.000	296.91	299.31	0.20732	0.20899	0.8796	115.6	114.7	112.4	111.5	0.02661	64.11	64.08
13	Pipe	5.000	296.98	299.38	0.20735	0.20901	0.8798	115.6	114.7	112.4	111.5	0.02686	64.22	64.19
14	Pipe	5.000	297.01	299.41	0.20736	0.20903	0.8799	115.6	114.7	112.4	111.5	0.02698	64.27	64.24
15	Pipe	5.000	296.98	299.38	0.20735	0.20901	0.8798	115.6	114.7	112.4	111.5	0.02686	64.22	64.19
16	Pipe	5.000	296.91	299.31	0.20732	0.20899	0.8796	115.6	114.7	112.4	111.5	0.02661	64.11	64.08

**Figure 4: Output window showing results for Natural Gas Burner Example, based on HX Heat Rate of 500 BTU/s**

By trial and error, the correct amount of heat that needs to be added to the system fluid from the heat exchanger to achieve a minimum stagnation temperature of 100 deg. F at each burner is about 998.8 Btu/sec.

## Conclusion

Using AFT Arrow, you were able to determine the amount of heat required by the heat exchange to supply natural gas to the burners at the minimum required temperature.

The goal seeking capabilities in the AFT Arrow GSC Module would allow this problem to be solved directly without manual iteration. The variable would be specified as the heat exchanger heat rate, and the goal would be the minimum discharge pipe exit temperature.

# Compressed Air System

## Summary

---

This example will demonstrate how to determine the range of outlet temperatures for a system given the range values for the system parameters.

## Topics Covered

- Modeling Compressors with curve data
- Entering heat transfer data for pipes
- Using Scenarios
- Using the Copy Data From Jct... editing feature

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

## Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Arrow installation:

- *US - Compressed Air System.aro*

## Problem Statement

In this example, four machine tools are supplied air for operations. The machine tools are sensitive to temperature; therefore, the manufacturer needs to know the delivery temperature extremes the tools will see.

The air is taken from atmospheric conditions outside the building. Atmospheric pressure will remain constant at 0 psig, and the air temperature outside can vary from 0 to 110 deg. F. The temperature inside the building ranges from 70 to 75 deg. F.

The compressor used to drive the system has the following pressure characteristics:

- 12 psid at 0 lbm/sec
- 10 psid at 0.5 lbm/sec
- 6 psid at 1 lbm/sec

The compressor efficiency is not known for certain, but it is expected to be between 80% and 90%.

The nozzles at the tools have a pressure drop of 8 psid at 0.2 lbm/sec. The nozzles discharge to atmospheric pressure.

The pipes in the system are uninsulated steel with an ambient air velocity that varies from 1 to 15 miles/hr. The pipe at the compressor inlet is heavily insulated (adiabatic). All of the components in the system are at an elevation of 0 feet.

What are the temperature extremes at the tools that this system could have?

### **Step 1. Start AFT Arrow**

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

### **Step 2. Define the Fluid Properties Group**

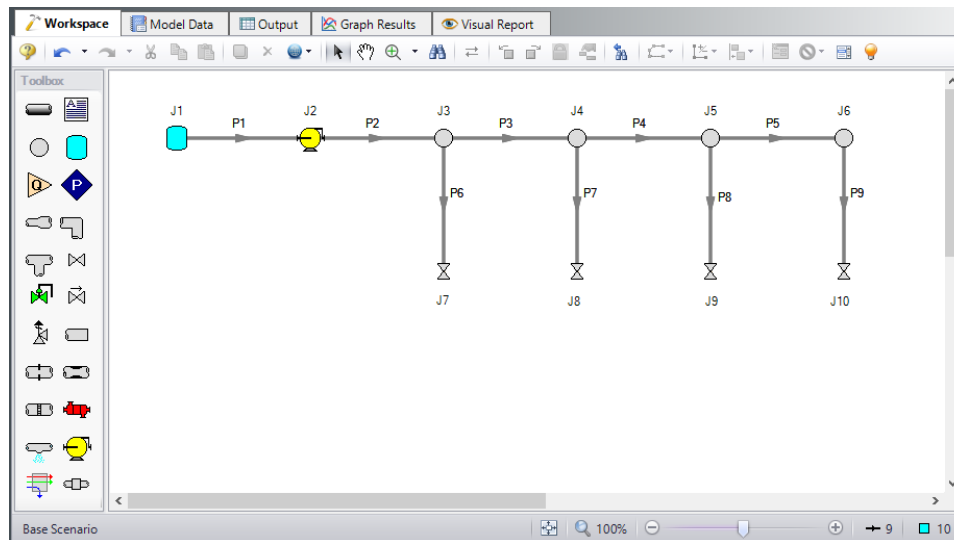
---

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 = Air
    - i. After selecting, click Add to Model
  - c. Equation of State = Redlich-Kwong
  - d. Enthalpy Model = Generalized
  - e. Specific Heat Ratio Source = Library

### **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 Compressed Air System Model**

For this problem, you are required to determine the temperature extremes at the tools based on the range of system specifications. To do this, you will use the Scenario Manager to examine the hot and cold extremes for the system.

The Scenario Manager is a powerful tool for managing variations of a model, referred to as scenarios.

The Scenario Manager allows you to:

- Create, name, and organize scenarios
- Select the scenario to appear in the Workspace (the 'current' scenario)
- Delete, clone, and rename scenarios
- Duplicate scenarios and save them as separate models
- Review the source of a scenario's specifications
- Pass changes from a parent scenario to its child scenarios

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 Base Scenario will contain all of the basic model data for the system.

## 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 (feet)
1	2 inch	1
2-5	2 inch	25
6-9	1 inch	10

### Junction Properties

- 1. J1 Tank
  - a. Elevation = 0 feet
  - b. Fluid = Air
  - c. Pressure = 0 psig
  - d. Temperature = 110 deg. F
- 2. J2 Compressor/Fan
  - a. Elevation = 0 feet
  - b. Model Type = Centrifugal Compressor
  - c. Compressor Model = Compressor Curve
  - d. Added Pressure = Stagnation
  - e. Compression Process Thermodynamics = Determine From Efficiency Data
  - f. Enter Curve Data =

Mass	Pressure	Efficiency
lbm/sec	psid	Percent
0	12	80
0.5	10	80
1	6	80

- g. Curve Fit Order = 2
- h. Click Generate Curve Fit Now

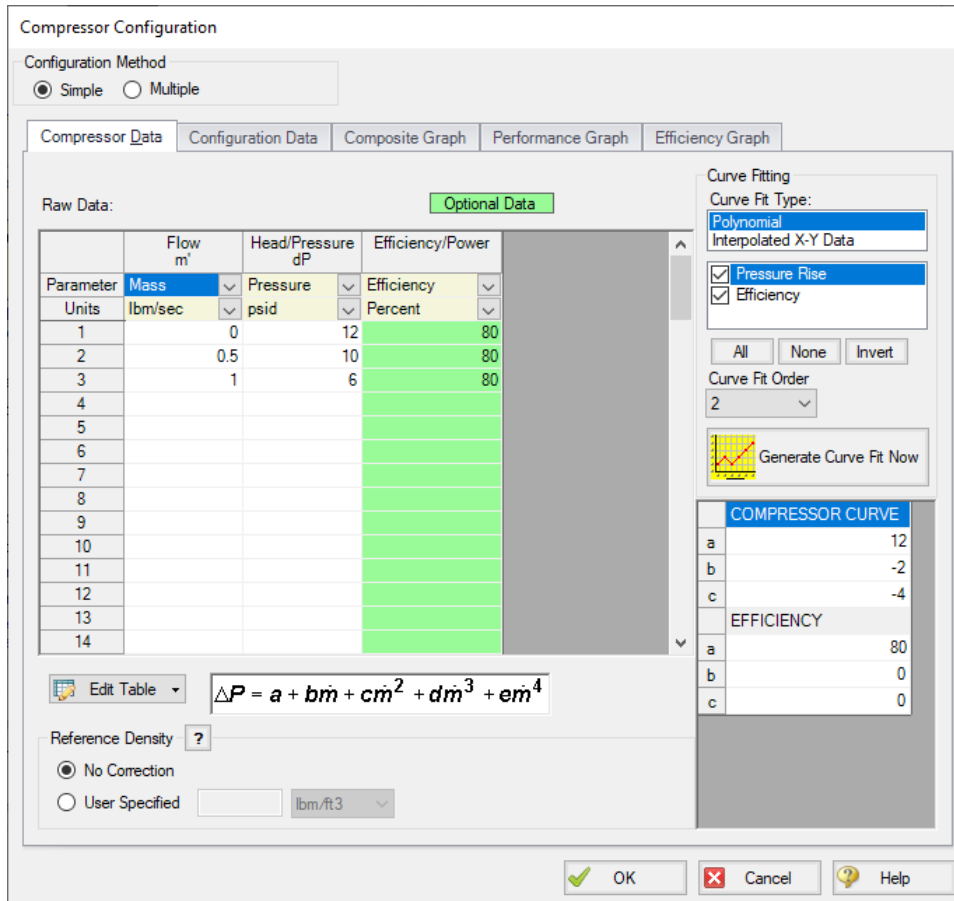


Figure 2: Compressor/Fan Configuration window

3. J3-J6 Branches
  - a. Elevation = 0 feet
4. J7 Valve
  - a. Elevation = 0 feet
  - b. Valve Data Source = User Specified
  - c. Subsonic Loss Model = Resistance Curve
  - d. Exit Valve (optional) = Checked
  - e. Exit Pressure = 0 psig
  - f. Exit Temperature = 70 deg. F
  - g. Enter Curve Data =

Mass	Pressure
lbm/sec	psid
0.2	8



- h. Click Fill As Quadratic
  - i. Curve Fit Order = 2
  - j. Click Generate Curve Fit Now
5. J8-J10 Valves
- a. Open the Valve Property window, and in the Copy Data From Jct drop-down list at the top, select J7. This will open the Copy Data From Junction window. Click All at the top. This will cause all of the properties to be set to the same as the properties for J7 (see Figure 3). Click OK. J8 now has all of the same properties as J7.
  - b. Repeat this process for J9 and J10.

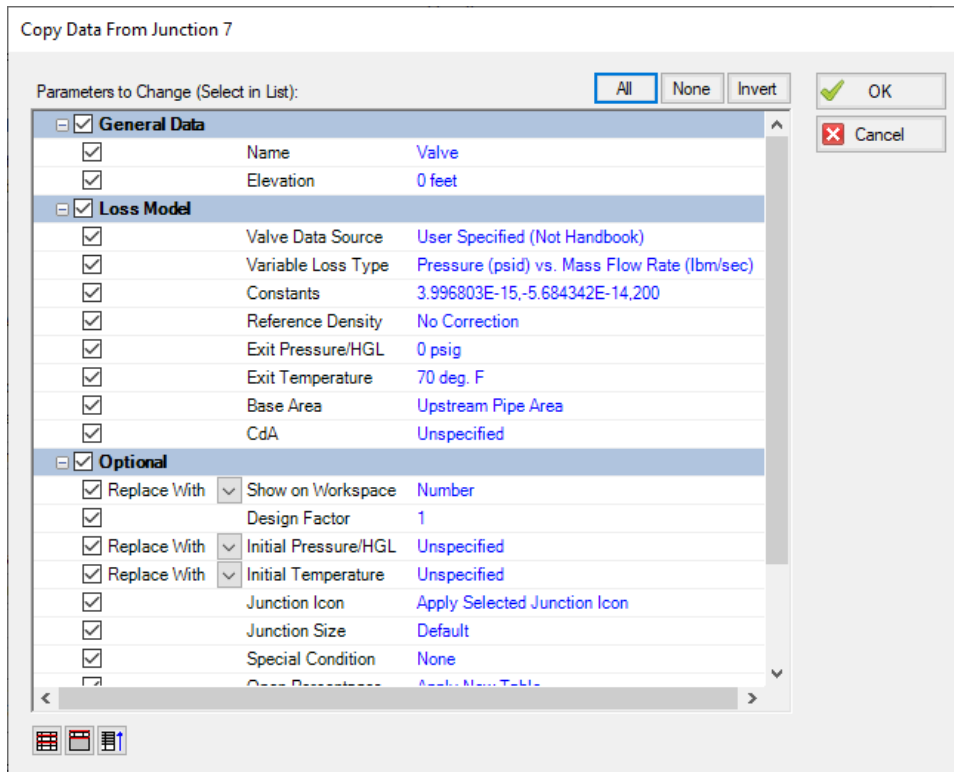



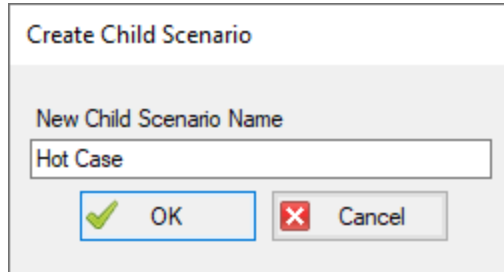
Figure 3: Copy Data From Junction 7 window for J8

➤ **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. Create the Scenarios

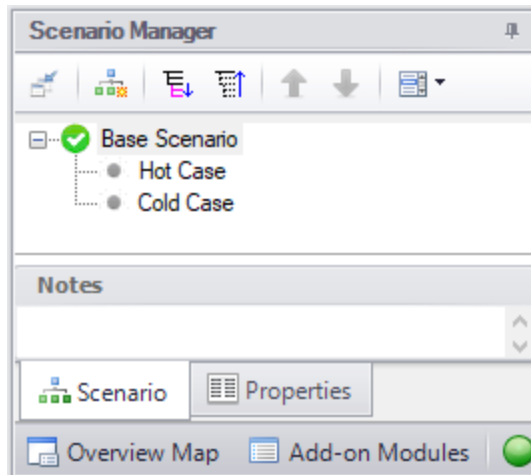
Now you will use the Scenario Manager on the Quick Access Panel to create our temperature extreme scenarios. Note that you can also use the Scenario Manager from the Tools menu if you prefer.

From the Quick Access Panel, either right click on the Base Scenario and select Create Child, or select the Create Child icon . Enter the name *Hot Case* in the Create Child Scenario window, as shown in Figure 4. Then click OK.



**Figure 4: New scenario named Hot Case in the Create Child Scenario window**

Select the Base Scenario in the Quick Access Panel, and then create another child. Enter the name *Cold Case* in the Create Child Scenario window and click OK. The Scenario Manager on the Quick Access Panel should now show two child scenarios below the Base Scenario, as shown in Figure 5.




**Figure 5: Scenario Manager with two child scenarios**

---

## Step 5. Modify the Hot Case Scenario

---

Double-click on the Hot Case scenario in the Quick Access Panel to load the Hot Case scenario. Alternatively, you can select the Hot Case scenario from the Quick Access Panel and then click on the Load Scenario icon  to load the scenario.

In order to simulate the Hot Case such that the tools would experience the highest temperatures, you must assume the highest inlet and ambient temperatures as well as the worst heat transfer properties of the piping and lowest compressor efficiency. Make these changes to the following pipes:

1. P2-P9 Pipes

- a. The heat transfer properties of the uninsulated pipes should be set to the values as shown in Figure 6 (the wall thickness for the Pipe Wall will vary, depending on the diameter of the pipe). Change the Heat Transfer Model to Convective Heat Transfer. Next, enter a Temperature of 75 deg. F and enter a Fluid Velocity of 1 miles/hr. Doing so will minimize the heat transfer occurring in the pipe, and result in the highest temperature at the tools. Use the Global Pipe Edit tool to define the heat transfer information for all pipes at once.

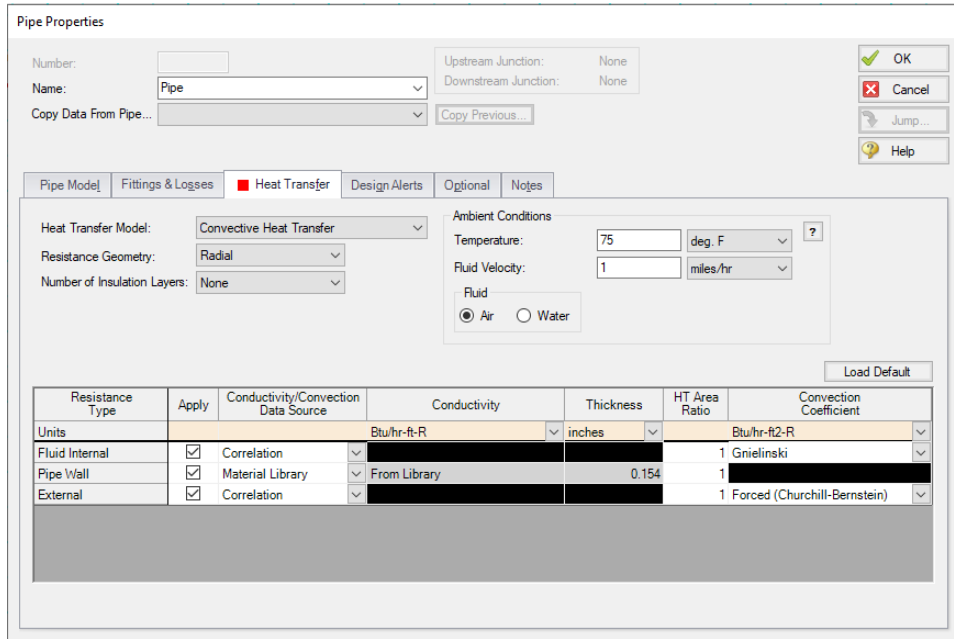


Figure 6: Pipe Heat Transfer Data for Hot Case Scenario

**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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

**Step 7. Examine the Output**

The Output window contains all the data that was specified in the Output Control window. The output for the Hot Case Scenario is shown in Figure 7. The valve output window shows the highest static temperature at the tools for the Hot Case is 196.24 deg. F at Tool 1 (T Static In).

Valve	Name	Mass Flow Rate Thru Jct (lbm/sec)	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)
7	Tool #1	0.1553	4.831	19.53	14.70	18.56	14.70	0.0	205.7	205.7	196.24
8	Tool #2	0.1415	4.009	18.70	14.70	17.89	14.70	0.0	196.6	196.6	188.36
9	Tool #3	0.1353	3.665	18.36	14.70	17.62	14.70	0.0	184.7	184.7	177.20
10	Tool #4	0.1342	3.614	18.31	14.70	17.60	14.70	0.0	165.5	165.5	158.47

Figure 7: Output window showing the results for Hot Case Scenario

## Step 8. Modify the Cold Case Scenario

Open the Scenario Manager again from the Tools menu (or use the Scenario Manager in the Quick Access Panel). Select the Cold Case scenario in the Scenario Manager, and then click Load As Current Scenario to open the Cold Case scenario.

In order to simulate the Cold Case such that the tools would experience the lowest temperatures, you must assume the lowest inlet and ambient temperatures as well as the best heat transfer properties of the piping and highest compressor efficiency. Make these changes to the following objects:

1. J1 Tank
  - a. Temperature = 0 deg. F
2. J2 Compressor
  - a. The compressor will generate less heat if it is more efficient. Since this is the lower temperature case, you must assume the best feasible efficiency, which is 90%. This data is entered in the Compressor/Fan Configuration window. Since you used 80% in the table when you added the compressor information in the Base Scenario (Step 6), change the efficiency in the Raw Data table to 90%, and then click Generate Curve Fit Now to generate a new compressor curve.
3. Pipes P2-P9
  - a. Set the heat transfer properties of the uninsulated pipes after the compressor to the values that will result in the lowest gas temperatures at the tools. The appropriate ambient temperature to use is the lowest of 70 deg. F. However, it is not clear which convection coefficient should be used. If the heat of compression at the compressor heats the gas to a temperature higher than 70 deg. F, we should use the highest ambient air velocity of 15 miles/hr. This will permit maximum cooling of the gas in the pipe. On the other hand, if the compressor discharge temperature is less than 70 deg. F, then the lowest ambient air velocity of 1 miles/hr should be used. This will yield the least heating of the gas in the pipes, giving the lowest possible temperature at the tools. Here we will use the value of 15 miles/hr. As just discussed, in practice one would need to verify the validity of this assumption.

## 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 Arrow solver converges on the answer. Once the solver has

converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

### Step 10. Examine the Output

The Output window contains all the data that was specified in the Output Control window. The output for the Cold Case Scenario is shown in Figure 8. The valve output window shows the lowest static temperature at the tools for the Cool Case is 66.09 deg. F at Tool 4 (Tstatic In).

Note that the compressor discharge stagnation temperature is 77.95 deg. F, and thus the assumption of 15 mph for the ambient air velocity was appropriate.

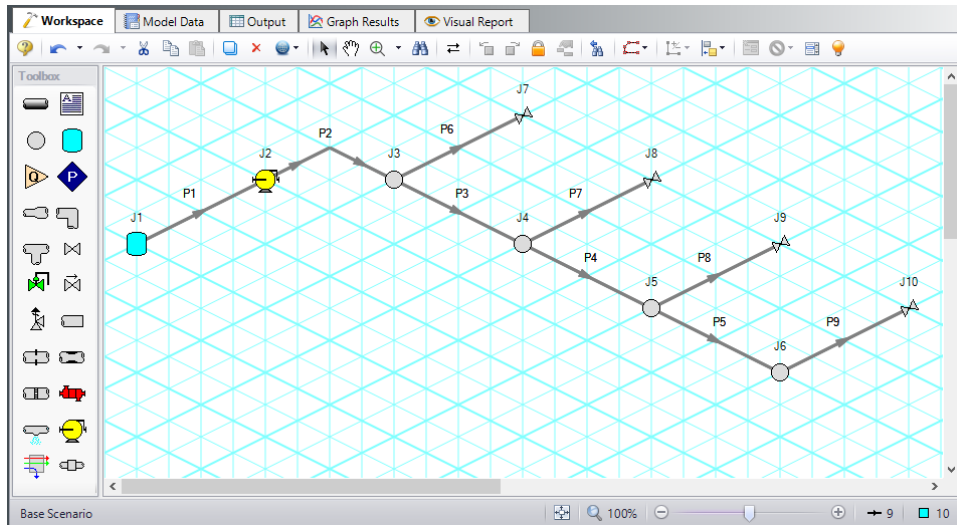
Valve	Name	Mass Flow Rate Thru Jct (lbm/sec)	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)
7	Valve	0.1624	5.281	19.98	14.70	19.16	14.70	0.000	75.32	75.32	68.94
8	Valve	0.1497	4.485	19.18	14.70	18.46	14.70	0.000	74.06	74.06	68.24
9	Valve	0.1437	4.134	18.83	14.70	18.16	14.70	0.000	72.81	72.81	67.28
10	Valve	0.1421	4.047	18.74	14.70	18.08	14.70	0.000	71.51	71.51	66.09

**Figure 8: Output window showing results for Cool Case Scenario**

### Step 11. Use the Isometric Grid

AFT Arrow 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 Arrow includes an Isometric Pipe Drawing Mode for these cases. The isometric grid has three gridlines that are offset by 60°, representing the x, y, and z axes. Figure 9 shows the compressed air system built on an isometric grid.

## Compressed Air System

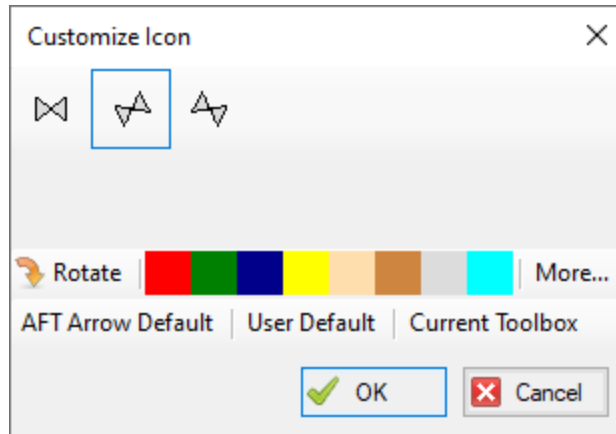


**Figure 9: Compressed air system with isometric grid shown**

You can enable the isometric grid by going to the Arrange menu. Under Pipe Drawing Mode, there are three options: 2D Freeform (default), 2D Orthogonal, and Isometric.

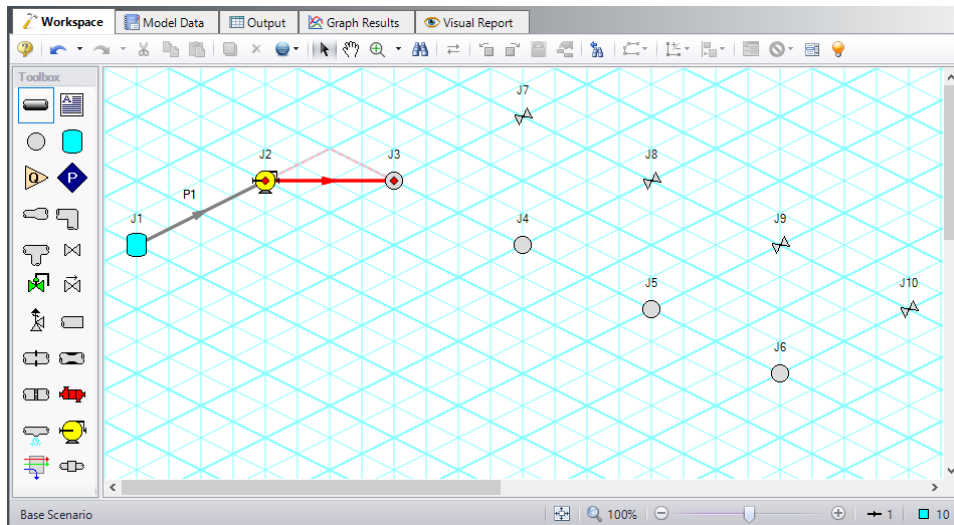
Creating the compressed air system on an isometric grid will demonstrate how to use this feature.

1. Go to the File menu and select New.
2. From the Arrange menu, Show Grid and choose Isometric under Pipe Drawing Mode.
3. Place junctions J1-J7 on the Workspace in the positions shown in Figure 9.
4. You will notice that placing junctions onto the Workspace follows the usual rules, however, the visual appearance of the icons can be more complex than on a 2D grid. Due to the increased number of axes, the preferred icon and rotation can be selected to obtain visual consistency. Right click on J7 and select Customize Icon. Click the Rotate button and select the icon shown in Figure 10.



**Figure 10:** Right click on the junction to open the *Customize Icon* window and select the preferred icon and rotation

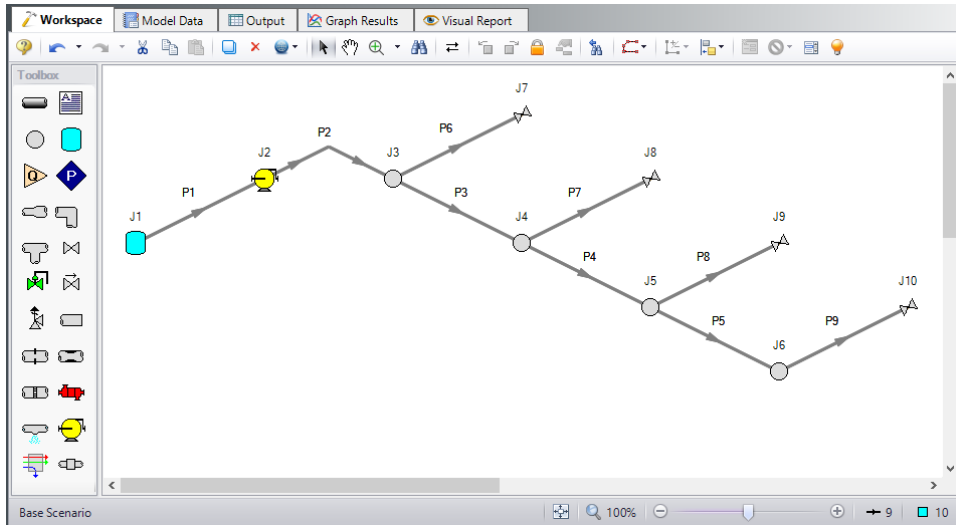
5. Copy J7 for the tools, J8-J10, to maintain the preferred icon.
6. Draw P1, as shown in Figure 11
7. Draw P2 from J2 to J3. A red-dashed preview line will show how the pipe will be drawn on the isometric grid. As you are drawing a pipe, you can change the preview line by clicking any arrow key on your keyboard or scrolling the scroll wheel on your mouse. Figure 11 shows Pipe 2 being drawn with the preview line.



**Figure 11:** A preview line shows when drawing or adjusting pipes

**Note:** You can hold the ALT key while adjusting a pipe by the endpoint. This can be used with the arrow key or mouse scroll wheel to change between different preview line options.

8. Draw pipes P3-P9, as shown in Figure 12.
9. The grid can be shown or turned off in the Arrange menu.



**Figure 12: Compressed air system with isometric grid not shown**

## Conclusion

By carefully selecting the input parameters from the specified system parameters, you were able to use AFT Arrow to determine the temperature extremes of the gas being supplied to the tools in the compressed air system. This information can now be sent to the tool manufacturer, who will use it to compensate for the temperature sensitivity of the tools.



# Process Steam System

## Summary

---

The objective of this example is to determine the minimum delivery pressure of a process steam supply system.

## Topics Covered

- Determining the minimum delivery pressure of a steam supply system.
- Understanding Solution Control sensitivity

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

## Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Arrow installation:

- ***US - Process Steam System.aro***

## Problem Statement

A process steam supply system has two boilers at 300 psia, and 800 deg. F.

All of the pipes in the system have 2 inches of insulation with a thermal conductivity of 0.02 Btu/hr-ft-R, and an ambient air velocity of 7 mph. The ambient temperature is 75 deg. F. The pipes are all Steel - ANSI STD (schedule 40). All of the elbows in the system are Smooth, with an r/D of 1.5.

All of the tees in the system should be modeled using the Detailed loss model, with sharp edges.

The system has six users with the following peak flow demands:

1. 40,000 lbm/hr
2. 50,000 lbm/hr
3. 60,000 lbm/hr
4. 55,000 lbm/hr

5. 40,000 lbm/hr
6. 60,000 lbm/hr

Determine the following:

- What is the minimum delivery (stagnation) pressure of the system, and at which user?

---

### **Step 1. Start AFT Arrow**

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

---

### **Step 2. Define the Fluid Properties Group**

---

1. Navigate to the Fluid panel in Analysis Setup
2. Define the Fluid panel with the following inputs
  - a. Fluid Library = AFT Standard
  - b. Fluid = Steam
    - i. After selecting, click Add to Model
  - c. Equation of State = Redlich-Kwong
  - d. Enthalpy Model = Generalized
  - e. Specific Heat Ratio Source = Library

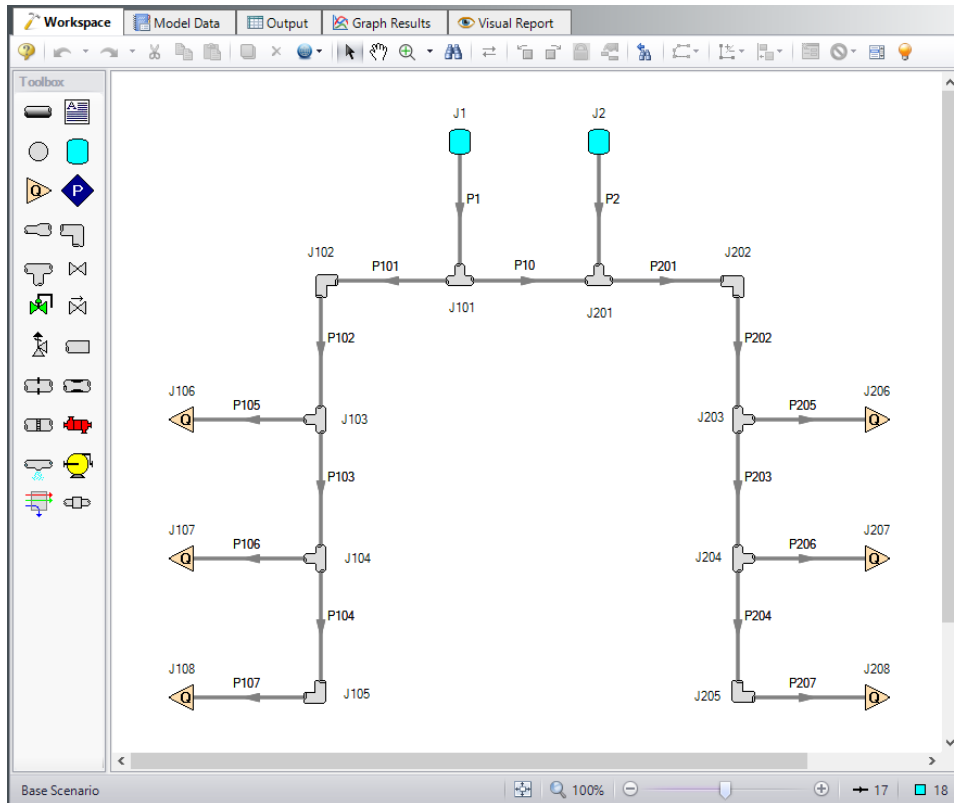
---

### **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.

# Process Steam System



**Figure 1: Model Layout for Process Steam 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.

## Pipe Properties

1. Pipe Model tab
  - a. Pipe Material = Steel - ANSI
  - b. Pipe Geometry = Cylindrical Pipe
  - c. Size = Use table below
  - d. Type = Use table below
  - e. Friction Model Data Set = Standard
  - f. Lengths = Use table below

Pipe	Size (inch)	Type	Length (feet)
1	12	STD	100
2	12	STD	150
10	12	STD	75

## Process Steam System

---

Pipe	Size (inch)	Type	Length (feet)
101	8	STD (schedule 40)	400
102-103	8	STD (schedule 40)	750
104	8	STD (schedule 40)	250
105	6	STD (schedule 40)	100
106	6	STD (schedule 40)	50
107	6	STD (schedule 40)	75
201	8	STD (schedule 40)	250
202-204	8	STD (schedule 40)	500
205 & 207	6	STD (schedule 40)	50
206	6	STD (schedule 40)	75

2. Heat Transfer tab
  - a. Heat Transfer Model = Convective Heat Transfer
  - b. Number of Insulation Layers = 1 External
  - c. Temperature = 75 deg. F
  - d. Fluid Velocity = 7 miles/hr
  - e. Insulation #1 Conductivity = 0.02 Btu/hr-ft-R
  - f. Insulation #1 Thickness = 2 inches

### Junction Properties

1. J1 & J2 Tanks
  - a. J1 Name = Boiler 1
  - b. J2 Name = Boiler 2
  - c. Elevation = 0 feet
  - d. Pressure = 300 psia
  - e. Temperature = 800 deg. F
2. All Bends

- a. Elevation = 0 feet
  - b. Type = Smooth Bend (flanged or butt-welded)
  - c. Angle = 90 Degrees
  - d.  $r/D = 1.5$
3. All Tee/Wye
- a. Elevation = 0 feet
  - b. Loss Model = Detailed
  - c. Type = Tee (Sharp, straight run)
  - d. Initial Pressure = 100 psig (on Optional tab)
4. All Assigned Flows
- a. Elevation = 0 feet
  - b. Type = Outflow
  - c. J106 Flow Rate = 40,000 lbm/hr
  - d. J107 Flow Rate = 50,000 lbm/hr
  - e. J108 Flow Rate = 60,000 lbm/hr
  - f. J206 Flow Rate = 55,000 lbm/hr
  - g. J207 Flow Rate = 40,000 lbm/hr
  - h. J208 Flow Rate = 60,000 lbm/hr

---

**Note:** Sometimes adding initial guesses to pipes or junctions can help difficult solutions converge.

---

➤ **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 Steady Solution Control Group**

---

There is an option available in Solution Control when using one of the marching methods that can sometimes result in an overall reduction in solution runtime. This option is to first solve the system using the Lumped Adiabatic method and then use these results as a starting point for the marching solution. Since the Lumped Adiabatic solution can typically be obtained much faster, this can provide an overall reduction in runtime for the marching method.

The Steady Solution Control Group is already defined using the default inputs. This means that no user input is required to run the model. However, the Lumped Adiabatic initialization will be used for this model.

To activate this option, do the following:

1. Open Analysis Setup
2. Navigate to the Solution Method panel in the Steady Solution Control group
3. Click the box to turn on First Use Lumped Adiabatic Method To Obtain Initial Stating Point For Marching Solution

The Steady Solution Control Group is now defined for this model.

### 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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

### Step 6. Examine the Output

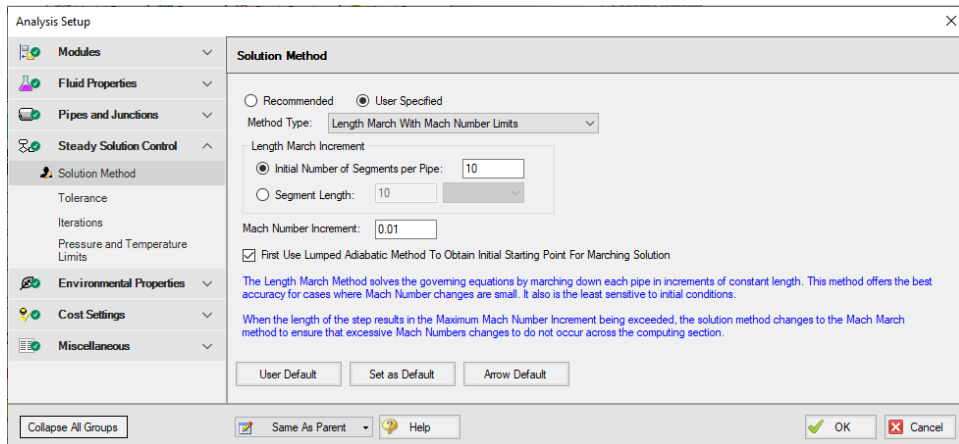
The pipes output section, shown in Figure 2, indicates the minimum delivery stagnation pressure is 56.77 psia at J108.

<span>⌵</span> All Junctions Multiple Losses Assigned Flow Bend Tank Tee or Wye							
Assigned Flow	Name	Mass Flow Rate Thru Jct (lbm/sec)	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)
106	Assigned Flow	11.11	0.0000	161.02	161.02	159.51	159.51
107	Assigned Flow	13.89	0.0000	77.46	77.46	72.17	72.17
108	Assigned Flow	16.67	0.0000	56.77	56.77	43.80	43.80
206	Assigned Flow	15.28	0.0000	206.35	206.35	204.10	204.10
207	Assigned Flow	11.11	0.0000	182.50	182.50	181.16	181.16
208	Assigned Flow	16.67	0.0000	172.74	172.74	169.54	169.54

Figure 2: Output window showing results for Process Steam System example

### Step 7. Solution Control Sensitivity

Open Analysis Setup and navigate to the Solution Method panel again. Change the Solution Method to User Specified with an Initial Number of Segments per Pipe as 10 (see Figure 3). Rerun the model, and compare the results to the previous run.



**Figure 3: The Solution Method panel can be used to adjust the solution sensitivity**

Repeat the run again with 20 segments, and 40 segments. What are the results?

- With 2 segments, the minimum (stagnation) pressure is 56.77 psia at J108.
- With 10 segments, the minimum (stagnation) pressure is 61.66 psia at J108.
- With 20 segments, the minimum (stagnation) pressure is 63.29 psia at J108.
- With 40 segments, the minimum (stagnation) pressure is 63.59 psia at J108.

This example points out the fact that some models are sensitive to the Solution Method used. While with most systems changing the solution control will not affect the results, in systems such as this where there are large pressure changes and high Mach numbers, the Solution Control settings can have a strong effect on the results. This is something that ALL engineers should be aware of, and keep in mind when creating computer model solutions.

# Hydrocarbon Relief System

## Summary

---

This example will demonstrate how to determine the relief capacity of a system, where sonic choking occurs, and the pressure drop across a shock wave.

## Topics Covered

- Understanding sonic choking
- Observing pressure drop across shock waves
- Using fluid mixtures

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

## Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Arrow installation:

- ***US - Hydrocarbon Relief System.aro***

## Problem Statement

A mixture of 89% methane and 11% hydrogen (by mass) exists at a process facility. There is a plan to increase the process conditions to 935 psig at 100 deg. F. The relief system needs to be re-evaluated to determine if it will still be adequate under these conditions. Assume that all of the pipes are well insulated (i.e. adiabatic).

Determine the following:

- What is the relief capacity of the system under the new conditions?
- Where does sonic choking occur in the system, and why?
- What is the pressure drop across the shock wave(s)?



### Step 1. Start AFT Arrow

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

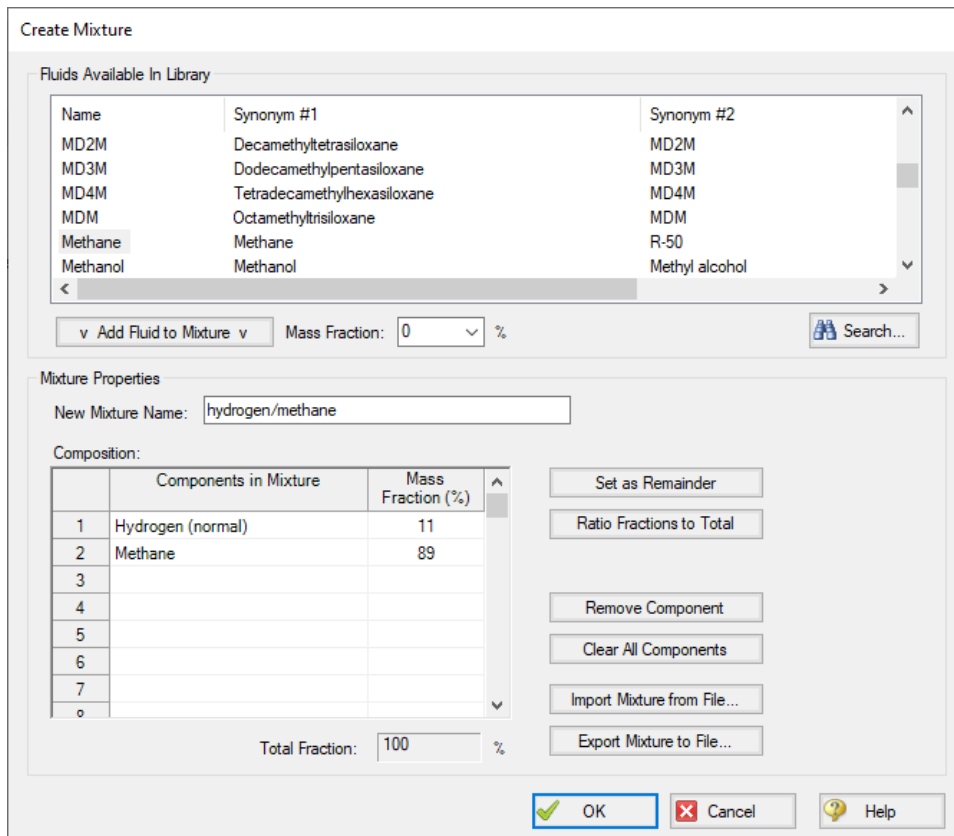
To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

### Step 2. Define the Fluid Properties Group

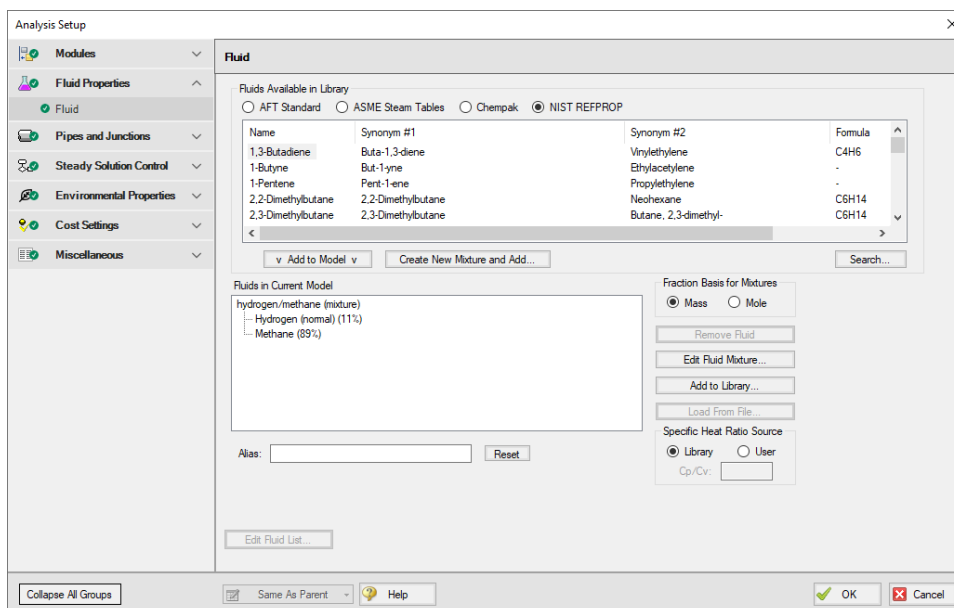
---

For this analysis, you will need to create and use a fluid mixture using the NIST REFPROP fluid library.

1. Open Analysis Setup from the toolbar or from the Analysis menu
2. Navigate to the Fluid panel
3. In the Fluids Available in Library section, select NIST REFPROP
4. Click Create New Mixture and Add to open the Create Mixture window
5. Enter the name *hydrogen/methane* in the New Mixture Name field
6. Select Hydrogen (normal) from the list of Fluids Available In Library
7. Enter a Mass Fraction of 11% in the Mass Fraction field, then click Add Fluid to Mixture to add hydrogen to the list components in the mixture
8. Repeat this process for Methane, with a Mass Fraction of 89%
9. The Create Mixture window should now appear as shown in Figure 1
10. Click OK to add the mixture to the Fluids in Current Model on the Fluid panel, as shown in Figure 2
11. Ensure that Library is selected as the Specific Heat Ratio Source



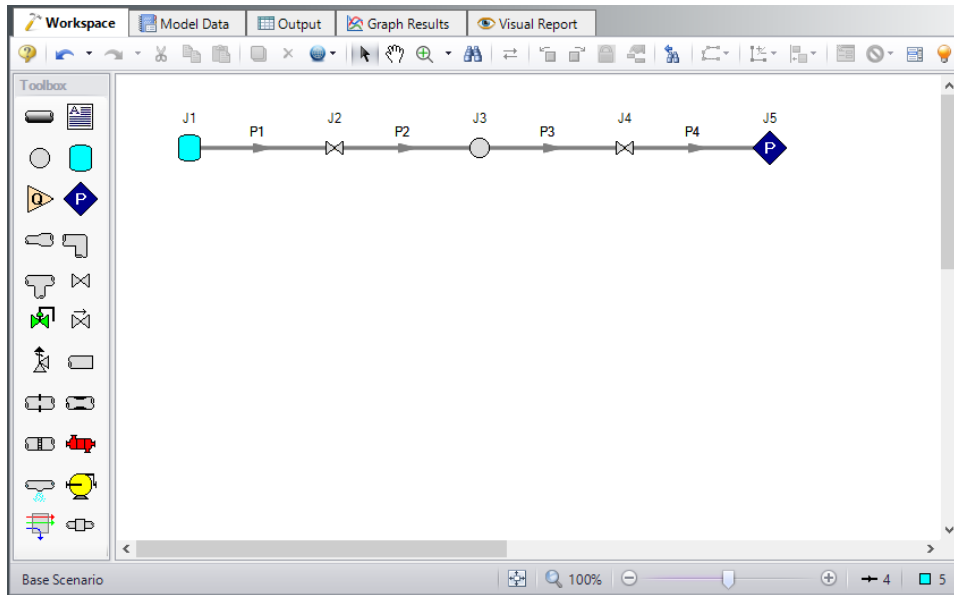
**Figure 1: The Create Mixture window is used to create mixtures of Refprop fluids**



**Figure 2: The Fluid panel in Analysis Setup**

## 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: Model layout for Hydrocarbon Process Relief System 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. Note the following:

- The Total K Factor listed for the pipes should be entered as the Total K Factor on the Fittings & Losses tab in the Pipe Properties window.
- The CdA values given for the valves are required for Sonic Choking calculations.

### Pipe Properties

1. Pipe Model tab
  - a. Pipe Material = Steel - ANSI
  - b. Pipe Geometry = Cylindrical Pipe
  - c. Size = Use table below
  - d. Type = Use table below
  - e. Friction Model Data Set = Standard
  - f. Lengths = Use table below
  - g. Total K Factor = Use table below

## Hydrocarbon Relief System

---

Pipe	Size (inch)	Type	Length (feet)	Total K Factor
1	6	STD (schedule 40)	24	4.4
2	6	STD (schedule 40)	21	4.2
3	10	STD (schedule 40)	25	2.1
4	16	STD (schedule 30)	258	8.9

### Junction Properties

1. J1 Tank
  - a. Name = Process
  - b. Elevation = 12 feet
  - c. Fluid = hydrogen/methane (mixture)
  - d. Pressure = 935 psig
  - e. Temperature = 100 deg. F
2. J2 Valve
  - a. Name = Manual Valve
  - b. Elevation = 6 feet
  - c. Valve Data Source = User Specified
  - d. Subsonic Loss Model = K Factor
  - e. K = 10
  - f. CdA (for Sonic Choking - optional) = 15 inches<sup>2</sup>
3. J3 Branch
  - a. Name = Header Connection
  - b. Elevation = 12 feet
4. J4 Valve
  - a. Name = Relief Valve (assumed open)
  - b. Elevation = 0 feet
  - c. Valve Data Source = User Specified
  - d. Subsonic Loss Model = K Factor
  - e. K = 8.3
  - f. CdA (for Sonic Choking - optional) = 45 inches<sup>2</sup>
5. J5 Assigned Pressure
  - a. Name = Atmosphere
  - b. Elevation = 22 feet
  - c. Fluid = hydrogen/methane (mixture)

- d. Pressure = 0 psig
- e. Temperature = 100 deg. F
- f. Pressure Specification = Stagnation

➤ **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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

### Step 5. Specify the Output Control

---

Open the Output Control window by selecting Output Control from the Toolbar or Tools menu.

Set the units for Mass Flow to lbm/hr.

### Step 6. Examine the Output

---

The output shows the capacity of the system is 383,788 lbm/hr, as shown in Figure 4.

The Sonic Choking tab in the General section of the Output window shows that the system chokes at J3, the Header Connection, and at J5, the discharge to atmosphere. The system chokes at J3 because of the expansion from 6-inch pipe to 10-inch pipe. The system chokes at J5 because of endpoint choking.

The pressure drop across the choke point is the same as the change in stagnation pressure. This can be seen in the Junction output section. The pressure drop across the choke point at J3 is 37.43 psid, and the pressure drop across the shock wave at J5 is 31.81 psid.

# Hydrocarbon Relief System

<span style="float: left;">⌵</span> <span style="float: right;">General</span> <span style="float: right;">Warnings</span> <span style="float: right;">Design Alerts</span> <span style="float: right;">Sonic Choking</span> <span style="float: right;">Valve Summary</span> <span style="float: right;">Tank Summary</span>														
Sonic Choking Exists at Junction 3 (Header Connection) Sonic Choking Exists at Junction 5 (Atmosphere)														
<span style="float: left;">⌵</span> <span style="float: right;">Pipes</span> <span style="float: right;">Axial Points</span> <span style="float: right;">Heat Transfer</span> <span style="float: right;">Composition</span>														
Pipe	Name	Mass Flow Rate (lbm/hr)	Vel. In (feet/sec)	Vel. Out (feet/sec)	Mach # In	Mach # Out	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)
1	Pipe	385,788	378.0	434.4	0.1819	0.2105	122.15	949.7	827.54	927.7	802.23	1.6368	100.00	98.36
2	Pipe	385,788	634.5	1,847.9	0.3127	0.9900	283.13	577.1	293.98	539.6	158.61	2.7798	94.85	92.07
3	Pipe	385,788	509.6	582.9	0.2536	0.2909	30.18	256.6	226.37	245.6	213.80	0.4645	89.26	88.80
4	Pipe	385,788	449.9	1,820.8	0.2247	0.9780	77.68	124.2	46.50	120.0	25.51	0.2030	86.92	86.72
<span style="float: left;">⌵</span> <span style="float: right;">All Junctions</span> <span style="float: right;">Assigned Pressure</span> <span style="float: right;">Branch</span> <span style="float: right;">Tank</span> <span style="float: right;">Valve</span>														
Jct	Name	Mass Flow Rate Thru Jct (lbm/sec)	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)		
1	Process	107.2	0.00	949.70	949.70	949.70	949.7	0.000	100.00	100.00	100.000	100.00		
2	Manual Valve	107.2	250.44	827.54	577.10	802.23	539.6	3.515	98.36	94.85	93.360	84.56		
3	Header Connection	107.2	37.43	293.98	256.55	158.61	245.6	0.000	92.07	92.07	7.341	85.70		
4	Relief Valve (assumed open)	107.2	102.19	226.37	124.18	213.80	120.0	1.872	88.80	86.92	80.545	82.00		
5	Atmosphere	107.2	31.81	46.50	14.70	25.51	N/A	-13.278	86.72	100.00	6.381	100.00		

**Figure 4: Output window showing results for Hydrocarbon Process Relief System example**

# Refinery Relief System

## Summary

---

This example will demonstrate how to evaluate an emergency relief system for relief capacity, and mass and mole fraction of a discharge mixture for an environmental impact assessment.

## Topics Covered

- Evaluating Relief Valves
- Using Scenarios
- Using Global Junction Edit Feature

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

## Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Arrow installation:

- ***US - Refinery Relief System.aro***

## Problem Statement

A new emergency relief system at an oil refinery is being considered, and the process calculations need to be evaluated.

The system provides relief to processes for methane, propane, and ethane. Each process is at 200 psig when the relief event occurs.

The relief valve CdA is 15 inches<sup>2</sup>. Assume K=0 since this system will choke. The elbow is a standard elbow, and the tees should all be modeled as simple tees. All of the pipes are STD (schedule 40) Steel - ANSI pipe, with standard roughness, and the flow can be assumed to be adiabatic.

The process engineer has evaluated the relief capacity at the minimum process temperature of 300 deg. F. Depending on the properties of the crude oil, all three processes can operate at up to 700 deg. F. The process engineer did not look at this case due to time constraints, but he thinks the higher temperatures

will not affect the relief capacity. The process engineer also neglected any elevation changes to simplify the calculations, even though the relief point is 120 feet above the discharge header.

Determine the following:

- The relief capacity (i.e. flow rate) of each process.
- The mass and mole fraction of the discharge mixture for the environmental impact assessment.
- Determine if neglecting the high temperature case and the elevation effects are acceptable.

### **Step 1. Start AFT Arrow**

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

### **Step 2. Define the Steady Solution Control Group**

---

There is an option available in Solution Control when using one of the marching methods that can sometimes result in an overall reduction in solution runtime. This option is to first solve the system using the Lumped Adiabatic method and then use these results as a starting point for the marching solution. Since the Lumped Adiabatic solution can typically be obtained much faster, this can provide an overall reduction in runtime for the marching method.

The Steady Solution Control Group is already defined using the default inputs. This means that no user input is required to run the model. However, the Lumped Adiabatic initialization will be used for this model.

To activate this option, do the following:

1. Open Analysis Setup
2. Navigate to the Solution Method panel in the Steady Solution Control group
3. Click the box to turn on First Use Lumped Adiabatic Method To Obtain Initial Starting Point For Marching Solution

The Steady Solution Control Group is now defined for this model.

### **Step 3. Define the Fluid Properties Group**

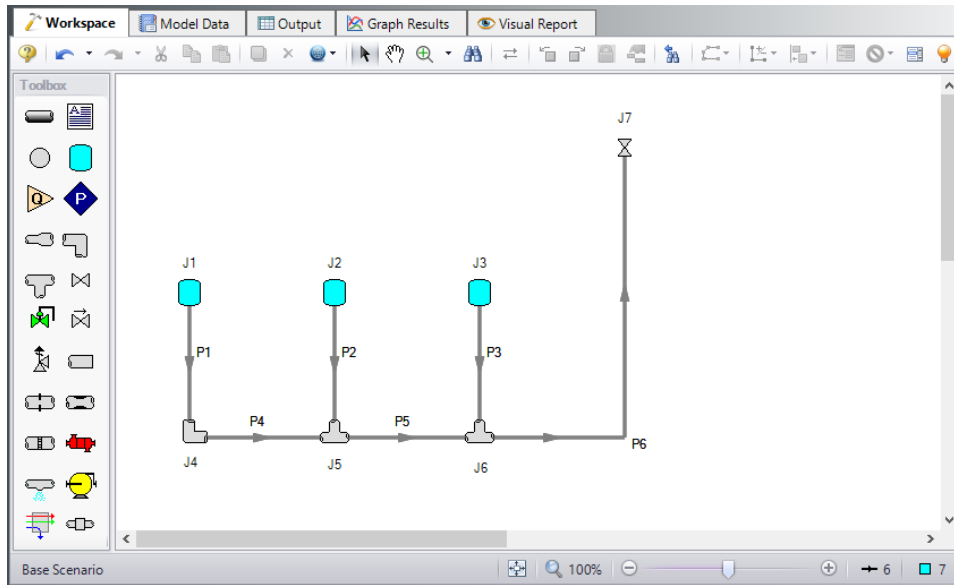
---

1. Open Analysis Setup from the toolbar or from the Analysis menu
2. Open the Fluid panel then define the fluid:
  - a. Fluid Library = NIST REFPROP
  - b. Fluid = Methane
    - i. After selecting, click Add to Model
  - c. Repeat this to add Ethane and Propane
  - d. Specific Heat Ratio Source = Library



## 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: Model layout for Refinery Relief System 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.

### 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 (inch)	Length (feet)
1-3	3	50

Pipe	Size (inch)	Length (feet)
4	3	25
5	4	25
6	6	150

## Junction Properties

1. All Tanks
  - a. J1 Name = Methane Tank
  - b. J2 Name = Ethane Tank
  - c. J3 Name = Propane Tank
  - d. Elevation = 0 feet
  - e. J1 Fluid = Methane
  - f. J2 Fluid = Ethane
  - g. J3 Fluid = Propane
  - h. Pressure = 200 psig
  - i. Temperature = 300 deg. F
2. J4 Bend
  - a. Elevation = 0 feet
  - b. Type = Standard Elbow (knee, threaded)
  - c. Angle = 90 Degrees
3. All Tees
  - a. Elevation = 0 feet
  - b. Loss Model = Simple
4. J7 Valve
  - a. Name = Relief Valve
  - b. Elevation = 0 feet
  - c. Valve Data Source = User Specified
  - d. Subsonic Loss Model = K Factor
  - e.  $K = 0$
  - f. CdA (for Sonic Choking - optional) = 15 inches<sup>2</sup>
  - g. Exit Valve (optional) = Checked
  - h. Exit Pressure = 0 psig
  - i. Exit Temperature = 70 deg. F

➤ **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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

## Step 6. Specify the Output Control

Open the Output Control window by selecting Output Control from the Toolbar or Tools menu.

With the Pipes button selected at the top, add Concentration Mass Fraction and Concentration Mole Fraction to the pipe output list by highlighting these parameters (holding CTRL will allow you to select both at the same time) and then clicking the purple Add arrow.

## Step 7. Examine the Output

The Composition tab in the pipes output section displays the component mass flow rate through each of the pipes, as shown in Figure 2).

The output for this model shows:

- The total relief capacity is 37.628 lbm/sec
  - 7.152 lbm/sec for methane
  - 12.55 lbm/sec for ethane
  - 17.93 lbm/sec for propane
- The mass fraction at discharge (observed from clicking on the Composition tab): 19.01% methane, 33.34% ethane, and 47.65% propane
- The mole fraction at discharge from the Composition tab: 35.11% methane, 32.86% ethane, and 32.02% propane

Pipe	Name	Mass Flow Rate Total (lbm/sec)	Mass Flow Rate Methane (lbm/sec)	Mass Flow Rate Ethane (lbm/sec)	Mass Flow Rate Propane (lbm/sec)	Conc. Mass Fraction Methane (Percent)	Conc. Mass Fraction Ethane (Percent)	Conc. Mass Fraction Propane (Percent)	Conc. Mole Fraction Methane (Percent)	Conc. Mole Fraction Ethane (Percent)	Conc. Mole Fraction Propane (Percent)
1	Pipe	7.152	7.152	0.00	0.00	100.00	0.00	0.00	100.00	0.00	0.00
2	Pipe	12.546	0.000	12.55	0.00	0.00	100.00	0.00	0.00	100.00	0.00
3	Pipe	17.928	0.000	0.00	17.93	0.00	0.00	100.00	0.00	0.00	100.00
4	Pipe	7.152	7.152	0.00	0.00	100.00	0.00	0.00	100.00	0.00	0.00
5	Pipe	19.698	7.152	12.55	0.00	36.31	63.69	0.00	51.65	48.35	0.00
6	Pipe	37.628	7.152	12.55	17.93	19.01	33.34	47.65	35.11	32.86	32.02

**Figure 2: Output window showing results for Refinery Relief System example**

## Step 8. Create a Child Scenario

---

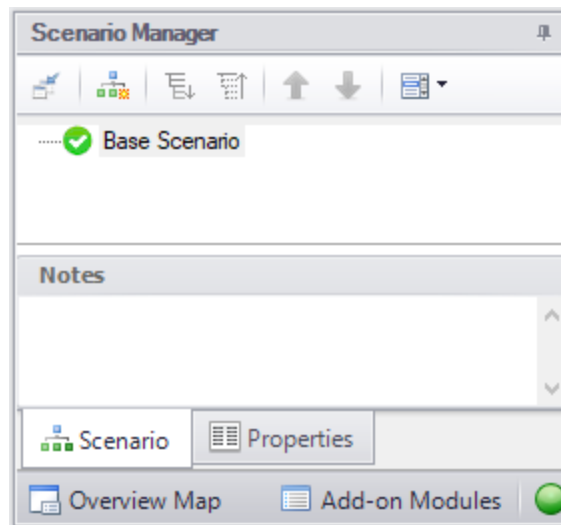
In order to determine if the process engineer has made good assumptions by neglecting the higher temperature process and the elevation data, it will be necessary to change the model. This can be done easily using the Scenario Manager on the Quick Access Panel. Note that the Scenario Manager can also be accessed from the Tools menu.

The Scenario Manager is a powerful tool for managing variations of a model, referred to as scenarios.

The Scenario Manager allows you to:

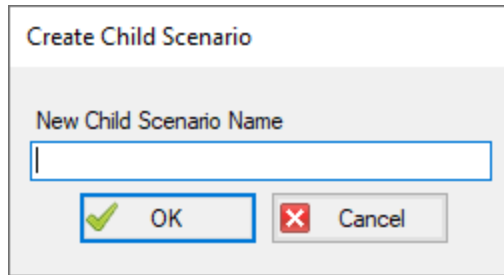
- Create, name and organize scenarios
- Select the scenario to appear in the Workspace (the 'current' scenario)
- Delete, copy and rename scenarios
- Duplicate scenarios and save them as separate models
- Review the source of a scenario's specifications
- Pass changes from a scenario to its variants

The Scenario Manager should currently appear as shown in Figure 3.

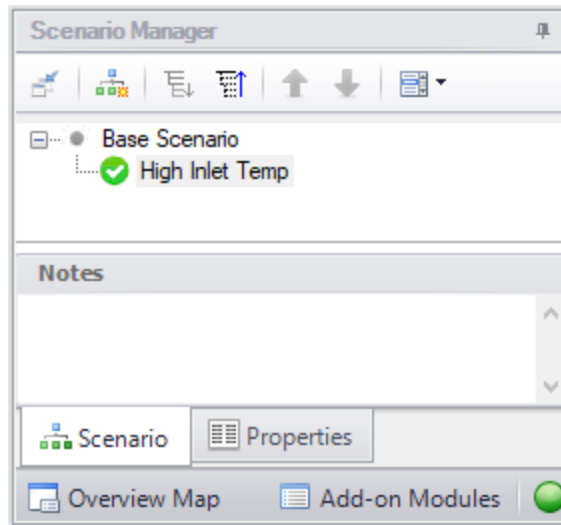


**Figure 3: The Scenario Manager window on the Quick Access Panel is used to create and manage model scenarios**

Create a child scenario by either right-clicking on the Base Scenario and then selecting Create Child. Enter the name *High Inlet Temp* in the Create Child Scenario window, as shown in Figure 4. Click OK. The new High Inlet Temp scenario should now appear in the Scenario Manager on the Quick Access Panel below the Base Scenario as shown in Figure 5.



**Figure 4: Create Child Scenario window**



**Figure 5: Scenario Manager on the Quick Access Panel with New Child Scenario**

---

## **Step 9. Modify the Higher Inlet Temp Scenario with Global Junction Editing**

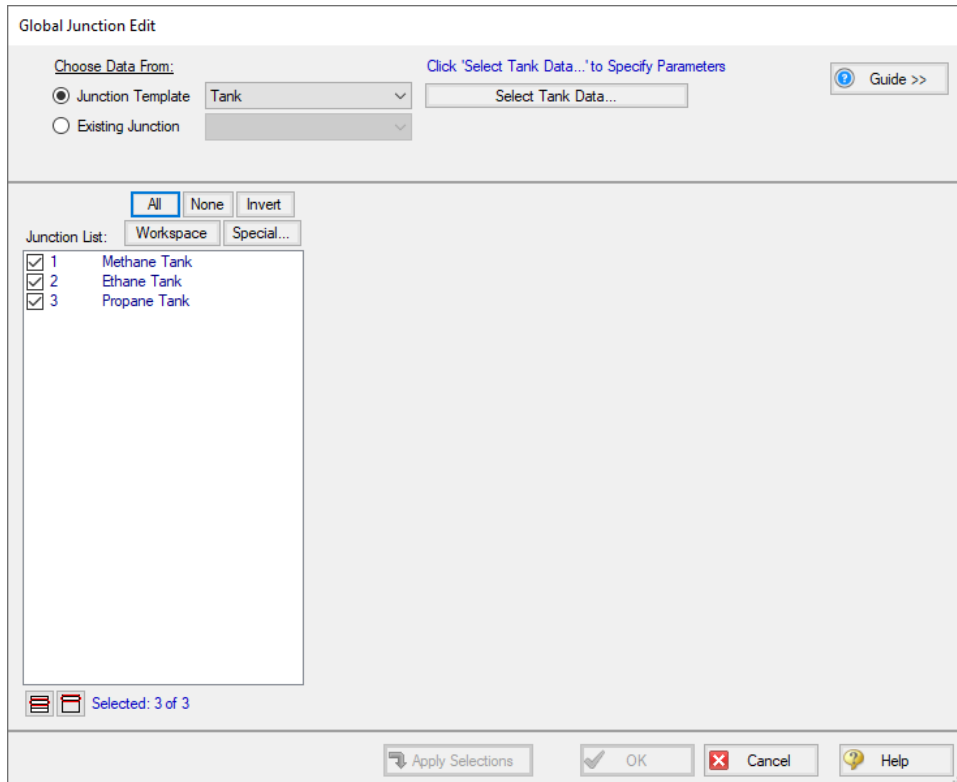
---

Double-click on the High Inlet Temp scenario in the Scenario Manager on the Quick Access Panel to make it the active scenario if it is not already.

To check the process engineer's assumption about the 700 deg. F process, the inlet temperature for all of the process tanks must be changed from 300 to 700 deg. F. These changes could be made by opening the Tank Properties window for each of the tanks and changing the inlet temperatures one at a time.

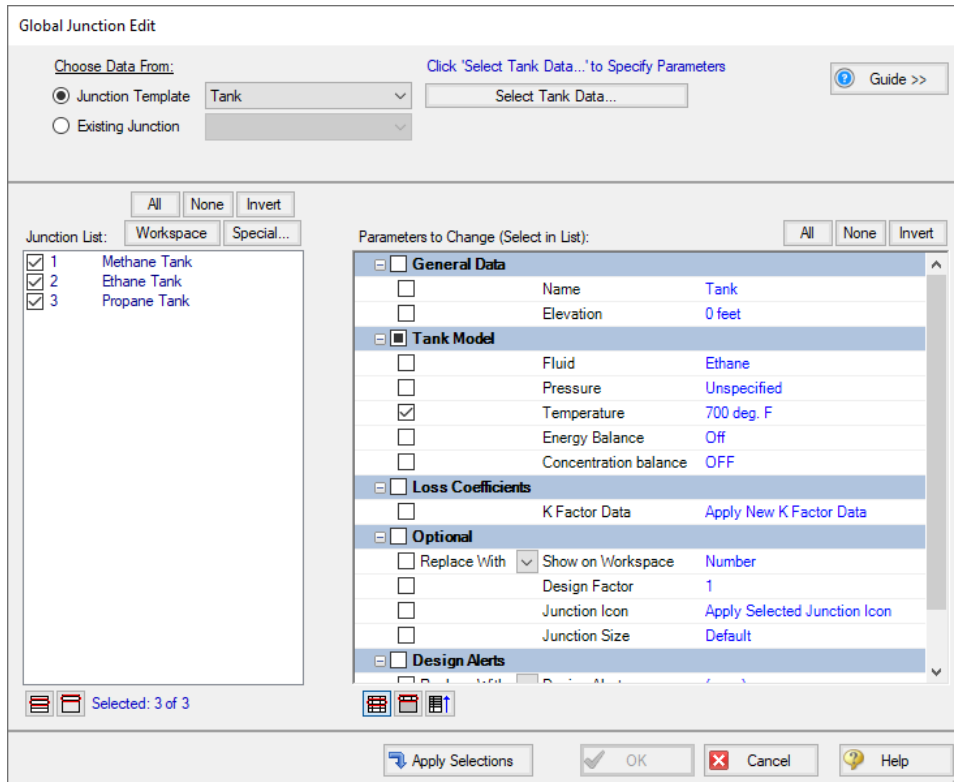
AFT Arrow has a feature that can be used to change the temperature for all of the tanks at the same time. This feature is called Global Junction Editing.

Open the Global Junction Edit window by selecting Global Junction Edit | Tanks from the Edit menu. Select all of the tanks in the Junction List by clicking All at the top (see Figure 6).



**Figure 6: Global Junction Edit window**

Click the Select Tank Data button. This will open a Tank Properties window. Enter a temperature of 700 deg. F and click OK to close the Tank Properties window. The Global Junction Edit window now contains a list of all the parameters that can be changed in the selected junctions. The data in the list is grouped together by the tab on the Properties window on which the data is entered. Any items that are selected in this list will be applied to all of the junctions selected in the Junction list. Check the box next to Temperature in the list, which now shows a value of 700 deg. F (see Figure 7). Since this is the only item in the parameter list that is selected, it is the only parameter that will be changed in the selected tank junctions. Click Apply Selections at the bottom then click OK to close the Global Junction Edit window. The temperatures for all three of the tanks have now been updated to 700 deg. F. Open the Properties window for one of the tanks to verify that the change was made.



**Figure 7: Global Junction Edit window with Temperature Parameter Selected**

## Step 10. 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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

## Step 11. Examine the Output

See Figure 8 for the output from this scenario. The output for this model shows:

At 700 deg. F, the relief capacity is reduced to 29.767 lbm/sec, a 21% reduction, which is significant. The original assumption and calculation by the process engineer based on this assumption was not conservative, as he assumed.

# Refinery Relief System

Pipe	Name	Mass Flow Rate Total (lbm/sec)	Mass Flow Rate Methane (lbm/sec)	Mass Flow Rate Ethane (lbm/sec)	Mass Flow Rate Propane (lbm/sec)	Conc. Mass Fraction Methane (Percent)	Conc. Mass Fraction Ethane (Percent)	Conc. Mass Fraction Propane (Percent)	Conc. Mole Fraction Methane (Percent)	Conc. Mole Fraction Ethane (Percent)	Conc. Mole Fraction Propane (Percent)
1	Pipe	5.732	5.732	0.000	0.00	100.00	0.00	0.00	100.00	0.00	0.00
2	Pipe	9.963	0.000	9.963	0.00	0.00	100.00	0.00	0.00	100.00	0.00
3	Pipe	14.068	0.000	0.000	14.07	0.00	0.00	100.00	0.00	0.00	100.00
4	Pipe	5.732	5.732	0.000	0.00	100.00	0.00	0.00	100.00	0.00	0.00
5	Pipe	15.695	5.732	9.963	0.00	36.52	63.48	0.00	51.89	48.11	0.00
6	Pipe	29.767	5.733	9.964	14.07	19.26	33.47	47.27	35.46	32.88	31.66

**Figure 8: Output window showing the results for High Inlet Temp Scenario**

## Step 12. Create and Run a Child Scenario for Elevation Effects

Create a child scenario to the High Inlet Temp scenario to examine the affect including the elevation information has on the model.

Change the elevation data for the J7 Valve to 120 feet. Rerun the model, and examine the results.

Adding the 120-foot elevation to the relief valve has a negligible effect on the system solution.



# Air Blower Sizing

## Summary

---

The objective of this example is to size a blower for an air distribution system, and evaluate the effect of different blower efficiencies on the blower size.

## Topics Covered

- Sizing a Blower
- Changing Compressor Efficiencies
- Using Design Factors
- Using Scenario Manager

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

## Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Arrow installation:

- *US - Air Blower Sizing.aro*

## Problem Statement

An air distribution system requires a new blower. All of the pipes in the system are STD (schedule 40) steel, with adiabatic flow. You are to use a design factor of 1.1 on all of the pipe and junction friction losses. You can neglect elevation changes, and any losses at the branches. All of the pipes leading to the discharge rooms discharge at atmospheric pressure.

Use Redlich-Kwong for the equation of state model, and Generalized for the enthalpy model.

For the blower, there are three compression process cases to be examined:

- Adiabatic
- Efficiency = 80 Percent (set as Nominal Efficiency)

- Efficiency = 60 Percent

For these three cases, determine:

- The size of the blower for both flow and pressure rise to deliver a minimum of 250 scfm to each room.
- How the different compression process models for the blower affect the blower sizing and the discharge temperatures.

### **Step 1. Start AFT Arrow**

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

### **Step 2. Define the Fluid Properties Group**

---

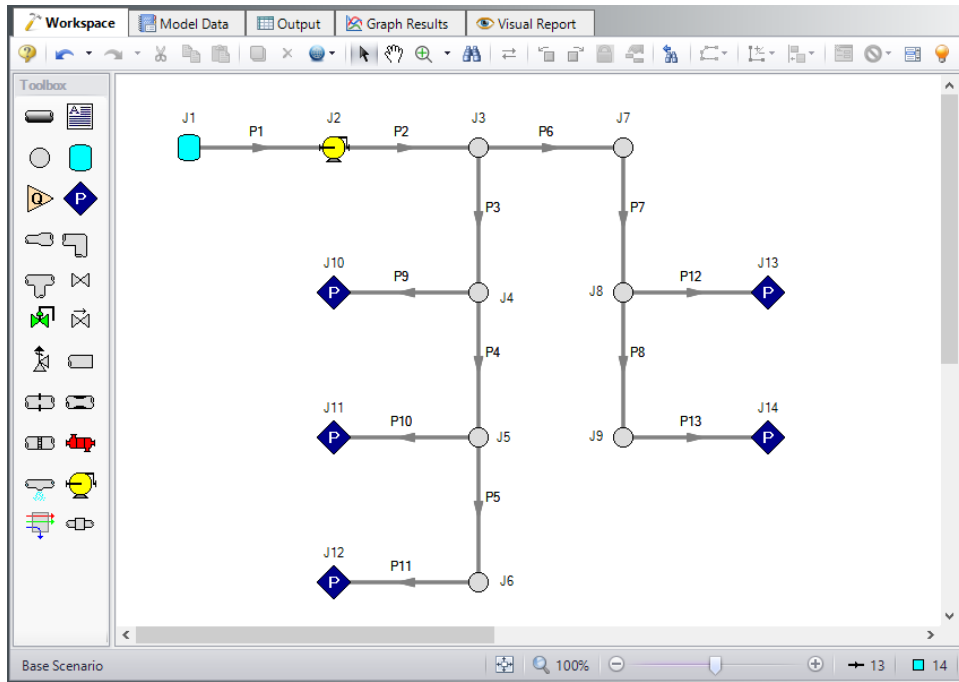
1. Open Analysis Setup from the toolbar or from the Analysis menu
2. Open the Fluid panel then define the fluid:
  - a. Fluid Library = AFT Standard
  - b. Fluid = Air
    - i. After selecting, click Add to Model
  - c. Equation of State = Redlich-Kwong
  - d. Enthalpy Model = Generalized
  - e. Specific Heat Ratio Source = Library

### **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.

# Air Blower Sizing



**Figure 1: Model layout for Air Blower Sizing 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.

## 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 (inch)	Length (feet)
1	6	40
2	6	100
3-5, 7-8	4	250
6	4	500
9-13	2	100

2. Optional tab
  - a. Pipe Friction = 1.1

### Junction Properties

For this example, we need to determine the correct flow through the compressor. Now we need to use our engineering judgment to take an initial guess at the flow rate through the compressor. Assume 1250 scfm to start:

1. J1 Tank
  - a. Elevation = 0 feet
  - b. Fluid = Air
  - c. Pressure = 0 psig
  - d. Temperature = 70 deg. F
2. J2 Compressor/Fan
  - a. Elevation = 0 feet
  - b. Compressor Type = Centrifugal Compressor
  - c. Compressor Model = Sizing
  - d. Compression Process Thermodynamics = Adiabatic
  - e. Compressor Sizing Parameter = Mass Flow Rate
  - f. Fixed Flow Rate = 1250 scfm
3. All Assigned Pressures
  - a. Elevation = 0 feet
  - b. Pressure Model tab
    - i. Fluid = Air
    - ii. Pressure = 0 psig
    - iii. Temperature = 70 deg. F
    - iv. Pressure Specification = Stagnation
  - c. Loss Coefficient tab
    - i. K - Flow into pipe = 1
    - ii. K - Flow out of pipe = 1
  - d. Optional tab
    - i. Junction Friction Loss = 1.1
4. All Branches
  - a. Elevation = 0 feet

➤ **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. Specify the Output Control**

---

Open the Output Control window by selecting Output Control from the Toolbar or Tools menu. Select the units scfm for the Mass Flow Rate parameter for both Pipes and Junctions output.

#### **Step 5. Create Child Scenarios**

---

Using the Scenario manager, create one child scenario named *1250 scfm* then three child scenarios to that scenario for the air blower evaluation. Name the first *Adiabatic*, name the second *80% Efficient*, and name the third *60% Efficient*. For the compressor in each scenario, open the Compressor/Fan Properties window, and set the following:

1. For the Adiabatic case, no changes are necessary. The Compression Process Thermodynamics for the Compressor is already set to Adiabatic and no Nominal Efficiency is entered.
2. For the 80% Efficient case, set the Nominal Efficiency to 80 Percent, and set the Compression Process Thermodynamics to Determine From Efficiency Data.
3. For the 60% Efficient case, set the Nominal Efficiency to 60 Percent, and set the Compression Process Thermodynamics to Determine From Efficiency Data.

#### **Step 6. Run the Models**

---

Load the Adiabatic scenario and 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 Arrow solver converges on the answer. This model runs very quickly. Now view the results by clicking the Output button at the bottom of the Solution Progress window.

Repeat this process for each of the three child scenarios.

#### **Step 7. Examine the Output**

---

The Output window contains all the data that was specified in the Output Control window.

Based upon the initial flow rate guess for the blower of 1250 scfm, not all rooms are receiving a minimum flow rate of 250 scfm. Therefore, the flow rate at the blower must be increased until each room receives a minimum flow rate of 250 scfm.

Determining the required blower flow rate can be easily accomplished through manual iteration on the blower flow rate. In order to automate this process of iteration, the AFT Arrow Goal Seek and Control (GSC) add-on module can assist with this greatly.

Right-click the 1250 scfm scenario and select Clone with Children. Iterate the blower mass flow rate until the 250 scfm is achieved.

The output for each of the scenarios is shown in Figure 2, Figure 3, and Figure 4. This output is based upon the final required blower flow rate that will deliver a minimum flow rate of 250 scfm to each room.

## Air Blower Sizing

The results are as follows:

Scenario	Total Flow	Stagnation Pressure Rise	Max Static Discharge Temp
	scfm	psid	deg. F
Adiabatic	1330	7.9476	136.48
80% Efficient	1330	8.1719	155.80
60% Efficient	1330	8.5650	189.95

Compressor/Fan Summary														
Jct	Results Diagram	Name	Mass Flow (lbm/sec)	Vol. Flow (ft <sup>3</sup> /sec)	dP Stag. (psid)	dP Static (psid)	dH Static (feet)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (hp)	Comp. Ratio Stag. (Percent)	BEP Q (ft <sup>3</sup> /sec)	BEP Mass (lbm/sec)	% of BEP
2	Show	Compressor/Fan	1.693	22.95	7.948	7.976	13.205	N/A	N/A	40.23	154.6	N/A	N/A	N/A

Pipes																
Pipe	Name	Mass Flow Rate (scfm)	Vel. In (feet/sec)	Vel. Out (feet/sec)	Mach # In	Mach # Out	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)
1	Pipe	1,330.0	113.19	114.38	0.10041	0.10147	0.1517	14.70	14.54	14.59	14.44	0.003540	70.00	70.00	68.93	68.90
2	Pipe	1,330.0	83.54	84.61	0.06968	0.07057	0.2814	22.49	22.21	22.42	22.13	0.003052	140.12	140.12	139.54	139.52
3	Pipe	904.1	116.35	129.75	0.09708	0.10829	2.2717	22.21	19.94	22.06	19.78	0.046875	140.12	140.07	138.99	138.67
4	Pipe	509.8	81.97	86.43	0.06837	0.07209	1.0240	19.94	18.91	19.87	18.85	0.021118	140.07	140.05	139.51	139.43
5	Pipe	250.0	42.30	42.93	0.03527	0.03579	0.2765	18.91	18.64	18.90	18.62	0.005676	140.05	140.04	139.90	139.89
6	Pipe	525.9	75.88	83.38	0.06328	0.06954	1.9890	22.21	20.22	22.15	20.15	0.040955	140.12	140.08	139.64	139.50
7	Pipe	525.9	83.38	88.07	0.06954	0.07345	1.0720	20.22	19.15	20.15	19.08	0.022034	140.08	140.05	139.50	139.41
8	Pipe	257.9	43.10	43.76	0.03593	0.03649	0.2895	19.15	18.86	19.13	18.84	0.005981	140.05	140.05	139.90	139.89
9	Pipe	294.3	181.15	240.40	0.15135	0.20120	4.7837	19.94	15.16	19.62	14.73	0.099243	140.07	139.97	137.34	135.17
10	Pipe	259.8	168.31	212.71	0.14059	0.17787	3.8601	18.91	15.05	18.66	14.73	0.080017	140.04	139.96	137.69	136.21
11	Pipe	250.0	164.28	204.79	0.13721	0.17122	3.6099	18.64	15.03	18.39	14.72	0.074768	140.04	139.96	137.80	136.48
12	Pipe	267.9	171.52	219.26	0.14328	0.18339	4.0722	19.15	15.08	18.88	14.73	0.084412	140.05	139.97	137.61	135.97
13	Pipe	257.9	167.54	211.15	0.13993	0.17657	3.8105	18.86	15.05	18.60	14.73	0.078979	140.04	139.97	137.71	136.26

Compressor/Fan														
Compressor/Fan	Name	Mass Flow Rate Thru Jct (scfm)	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)	dH Stag. (Btu/lbm)	H Stag. In (Btu/lbm)
2	Compressor/Fan	1,330.0	7.9476	14.54	22.49	14.44	22.42	-70.12	70.00	140.12	68.90	139.54	-16.79	195.1

**Figure 2: Output window showing the results for Adiabatic scenario**

# Air Blower Sizing

Compressor/Fan Summary														
Jct	Results Diagram	Name	Mass Flow (lbm/sec)	Vol. Flow (ft <sup>3</sup> /sec)	dP Stag. (psid)	dP Static (psid)	dH Static (feet)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (hp)	Comp. Ratio Stag. (Percent)	BEP Q (ft <sup>3</sup> /sec)	BEP Mass (lbm/sec)	% of BEP (Percent)
2	Show	Compressor/Fan	1.693	22.95	8.172	8.198	13.745	80.00	N/A	51.50	156.2	N/A	N/A	N/A

Pipes																
Pipe	Name	Mass Flow Rate (scfm)	Vel. In (feet/sec)	Vel. Out (feet/sec)	Mach # In	Mach # Out	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)
1	Pipe	1,330.0	113.19	114.38	0.10041	0.10147	0.1517	14.70	14.54	14.59	14.44	0.003540	70.00	70.00	68.93	68.90
2	Pipe	1,330.0	85.42	86.53	0.07013	0.07104	0.2884	22.72	22.43	22.64	22.35	0.005493	159.66	159.65	159.05	159.03
3	Pipe	804.2	118.99	132.94	0.09774	0.10922	2.3301	22.43	20.10	22.28	19.93	0.044312	159.65	159.61	158.48	158.14
4	Pipe	509.8	83.98	88.65	0.06895	0.07278	1.0523	20.10	19.05	20.03	18.98	0.019958	159.61	159.59	159.02	158.94
5	Pipe	250.0	43.38	44.04	0.03560	0.03614	0.2846	19.05	18.76	19.03	18.74	0.005371	159.58	159.58	159.43	159.42
6	Pipe	525.8	77.60	85.40	0.06370	0.07011	2.0404	22.43	20.39	22.36	20.32	0.038757	159.65	159.61	159.15	159.01
7	Pipe	525.8	85.40	90.30	0.07011	0.07414	1.1012	20.39	19.29	20.32	19.21	0.020874	159.61	159.59	159.01	158.92
8	Pipe	257.9	44.19	44.88	0.03627	0.03683	0.2978	19.29	18.99	19.27	18.97	0.005615	159.59	159.59	159.43	159.42
9	Pipe	294.3	185.64	248.19	0.15268	0.20450	4.9276	20.10	15.17	19.77	14.73	0.094177	159.60	159.51	156.75	154.40
10	Pipe	258.8	172.65	219.60	0.14196	0.18078	3.9791	19.05	15.07	18.78	14.73	0.075989	159.58	159.50	157.11	155.51
11	Pipe	250.0	168.57	211.43	0.13859	0.17401	3.7217	18.76	15.04	18.51	14.73	0.071106	159.58	159.50	157.22	155.80
12	Pipe	267.9	175.90	226.35	0.14464	0.18638	4.1965	19.29	15.09	19.01	14.73	0.080200	159.59	159.51	157.02	155.26
13	Pipe	257.9	171.86	217.98	0.14130	0.17944	3.9275	18.99	15.06	18.73	14.73	0.075012	159.58	159.51	157.13	155.57

Compressor/Fan														
Compressor/Fan	Name	Mass Flow Rate Thru Jct (scfm)	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)	dH Stag. (Btu/lbm)	H Stag. In (Btu/lbm)
2	Compressor/Fan	1,330.0	-8.1719	14.54	22.72	14.44	22.64	-89.66	70.00	159.66	68.90	159.05	-21.50	195.1

**Figure 3: Output window showing the results for 80% Efficient scenario**

Compressor/Fan Summary														
Jct	Results Diagram	Name	Mass Flow (lbm/sec)	Vol. Flow (ft <sup>3</sup> /sec)	dP Stag. (psid)	dP Static (psid)	dH Static (feet)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (hp)	Comp. Ratio Stag. (Percent)	BEP Q (ft <sup>3</sup> /sec)	BEP Mass (lbm/sec)	% of BEP (Percent)
2	Show	Compressor/Fan	1.693	22.95	8.565	8.589	14.726	60.00	N/A	71.49	158.9	N/A	N/A	N/A

Pipes																
Pipe	Name	Mass Flow Rate (scfm)	Vel. In (feet/sec)	Vel. Out (feet/sec)	Mach # In	Mach # Out	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)
1	Pipe	1,330.0	113.19	114.38	0.10041	0.10147	0.1517	14.70	14.54	14.59	14.44	0.003540	70.00	70.00	68.93	68.90
2	Pipe	1,330.0	88.67	89.85	0.07088	0.07182	0.3006	23.11	22.81	23.03	22.73	0.004944	194.22	194.21	193.57	193.54
3	Pipe	804.2	123.57	138.48	0.09882	0.11078	2.4317	22.81	20.38	22.65	20.20	0.040100	194.21	194.17	192.95	192.58
4	Pipe	509.8	87.47	92.50	0.06992	0.07394	1.1016	20.38	19.28	20.31	19.20	0.018127	194.17	194.15	193.54	193.44
5	Pipe	250.0	45.26	45.97	0.03617	0.03674	0.2987	19.28	18.98	19.26	18.96	0.004883	194.14	194.14	193.97	193.96
6	Pipe	525.8	80.57	88.90	0.06440	0.07106	2.1299	22.81	20.68	22.74	20.61	0.035034	194.21	194.18	193.67	193.52
7	Pipe	525.8	88.90	94.17	0.07106	0.07528	1.1521	20.68	19.53	20.61	19.45	0.018982	194.18	194.16	193.52	193.42
8	Pipe	257.8	46.08	46.83	0.03682	0.03742	0.3124	19.53	19.21	19.51	19.20	0.005127	194.15	194.15	193.98	193.97
9	Pipe	294.4	193.43	261.95	0.15490	0.21020	5.1805	20.38	15.20	20.04	14.74	0.085693	194.17	194.08	191.07	188.41
10	Pipe	258.8	180.20	231.79	0.14427	0.18582	4.1885	19.28	15.09	19.00	14.73	0.069336	194.14	194.07	191.46	189.63
11	Pipe	250.0	176.02	223.15	0.14091	0.17896	3.9186	18.98	15.06	18.72	14.73	0.064819	194.14	194.07	191.57	189.95
12	Pipe	267.9	183.51	238.89	0.14693	0.19156	4.4152	19.53	15.11	19.24	14.73	0.073059	194.15	194.08	191.37	189.36
13	Pipe	257.8	179.37	230.03	0.14360	0.18440	4.1332	19.21	15.08	18.94	14.73	0.068359	194.15	194.08	191.49	189.70

Compressor/Fan														
Compressor/Fan	Name	Mass Flow Rate Thru Jct (scfm)	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)	dH Stag. (Btu/lbm)	H Stag. In (Btu/lbm)
2	Compressor/Fan	1,330.0	-8.5650	14.54	23.11	14.44	23.03	-124.2	70.00	194.22	68.90	193.57	-29.85	195.1

**Figure 4: Output window showing the results for 60% Efficient scenario**

## Step 8. Evaluate 60% Efficiency with Heat Transfer

The engineers on the project have realized that the pipes are not insulated, and that the adiabatic assumption may be overly conservative. One engineer has also expressed concern that the discharge

## Air Blower Sizing

air could be too hot.

Create a child scenario to the 1330 scfm 60% efficient case named *With Heat Transfer*. Add heat transfer to the model on all the pipes by selecting Convective Heat Transfer as the Heat Transfer Model. For the Ambient Conditions, enter a Temperature of 80 deg. F, a Fluid Velocity of 7 miles/hr, and Air as the Fluid. See Figure 5 for this input.

Pipe Properties

Number: 1 Upstream Junction: 1  
Name: Pipe Downstream Junction: 2  
Copy Data From Pipe... Copy Previous...

Heat Transfer Model: Convective Heat Transfer  
Resistance Geometry: Radial  
Number of Insulation Layers: None

Ambient Conditions  
Temperature: 80 deg. F  
Fluid Velocity: 7 miles/hr  
Fluid:  Air  Water

Load Default

Resistance Type	Apply	Conductivity/Convection Data Source	Conductivity	Thickness	HT Area Ratio	Convection Coefficient
Units			Btu/hr-ft-R	inches		Btu/hr-ft <sup>2</sup> -R
Fluid Internal	<input checked="" type="checkbox"/>	Correlation			1	Gnielinski
Pipe Wall	<input checked="" type="checkbox"/>	Material Library	From Library	0.28	1	
External	<input checked="" type="checkbox"/>	Correlation			1	Forced (Churchill-Bernstein)

Figure 5: Pipe heat transfer window

Run the new scenario and determine the following:

- Do the compressor size requirements change significantly?
- How much do the discharge temperatures decrease?

### Step 9. Examine the Output for 60% Efficiency with Heat Transfer

The output for this scenario is shown in Figure 6.

The blower stagnation pressure rise decreases from 8.5650 to 7.4769 psid, which is a 13% reduction. The max discharge temperature decreases to 88.47 deg. F.



# Air Blower Sizing

Compressor/Fan Summary														
Jct	Results Diagram	Name	Mass Flow (lbm/sec)	Vol. Flow (ft <sup>3</sup> /sec)	dP Stag. (psid)	dP Static (psid)	dH Static (feet)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (hp)	Comp. Ratio Stag. (Percent)	BEP Q (ft <sup>3</sup> /sec)	BEP Mass (lbm/sec)	% of BEP
2	Show	Compressor/Fan	1.693	23.00	7.477	7.498	13.068	60.00	N/A	63.73	151.4	N/A	N/A	N/A

Pipes																
Pipe	Name	Mass Flow Rate (scfm)	Vel. In (feet/sec)	Vel. Out (feet/sec)	Mach # In	Mach # Out	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)
1	Pipe	1.330.0	113.19	114.63	0.10041	0.10159	0.1521	14.70	14.54	14.59	14.44	-1.1310	70.00	71.13	68.93	70.03
2	Pipe	1.330.0	91.32	88.81	0.07368	0.07313	0.2990	22.02	21.72	21.94	21.64	26.0145	181.93	155.92	181.24	155.26
3	Pipe	799.9	121.50	124.93	0.10010	0.10716	2.2413	21.72	19.48	21.57	19.32	48.2517	155.91	107.66	154.69	106.36
4	Pipe	509.5	79.30	80.25	0.06798	0.07012	0.9586	19.48	18.52	19.42	18.46	21.5176	107.66	86.15	107.14	85.61
5	Pipe	250.3	39.35	39.47	0.03437	0.03466	0.2528	18.52	18.27	18.51	18.25	5.7216	86.15	80.42	86.02	80.29
6	Pipe	530.1	80.29	77.88	0.06611	0.06817	1.9399	21.72	19.78	21.66	19.72	71.7316	155.92	84.18	155.38	83.68
7	Pipe	530.1	77.88	81.54	0.06817	0.07158	0.9959	19.78	18.79	19.72	18.72	3.2267	84.18	80.96	83.68	80.40
8	Pipe	260.1	39.92	40.43	0.03503	0.03551	0.2668	18.79	18.52	18.77	18.50	0.8903	80.96	80.07	80.82	79.93
9	Pipe	290.4	172.96	218.59	0.14851	0.19047	4.3727	19.48	15.11	19.18	14.73	15.2070	107.66	92.45	105.17	88.47
10	Pipe	259.2	155.94	191.98	0.13644	0.16867	3.5030	18.52	15.02	18.28	14.72	3.6612	86.14	82.48	84.12	79.41
11	Pipe	250.3	150.98	184.65	0.13277	0.16255	3.2731	18.27	15.00	18.05	14.72	0.3176	80.42	80.10	78.52	77.26
12	Pipe	270.0	158.70	199.07	0.13952	0.17528	3.7412	18.79	15.04	18.53	14.73	0.6205	80.95	80.33	78.85	77.03
13	Pipe	260.1	154.72	191.70	0.13611	0.16881	3.4996	18.52	15.02	18.28	14.72	0.1078	80.06	79.96	78.07	76.89

Compressor/Fan														
Compressor/Fan	Name	Mass Flow Rate Thru Jct (scfm)	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)	dH Stag. In (Btu/lbm)	H Stag. In (Btu/lbm)
2	Compressor/Fan	1.330.0	7.4769	14.54	22.02	14.44	21.94	-110.799	71.13	181.93	70.03	181.24	-26.61	195.4

Figure 6: Output window showing the results for 60% Efficiency Scenario with Heat Transfer in Pipes

# Compressed Air System - Cost

## Summary

---

This example demonstrates how to use the cost calculation features in AFT Arrow. All cost calculation features are included with Arrow.

## 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: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

## Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Arrow installation:

- ***US - Compressed Air System.aro***
- ***US - Compressed Air System Costs.aro***
- ***Compressed Air System.dat*** - engineering library
- ***Compressed Air System Costs.cst*** - cost library for ***Compressed Air System.dat***
- ***Steel - ANSI Pipe Costs.cst*** - cost library for **Steel - ANSI**

## Problem Statement

Use the cost calculation features in AFT Arrow to determine the cost of the compressed air system over a 10-year period for both the hot and cold cases. Include material, installation, and energy costs.

---

## Step 1. Start AFT Arrow

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

---

**Important:** The GSC module and cost calculations can be used in the same run, but we are not going to do this here. Before proceeding, make sure to disable GSC by selecting the radio button for Ignore in the Modules panel in Analysis Setup.

---

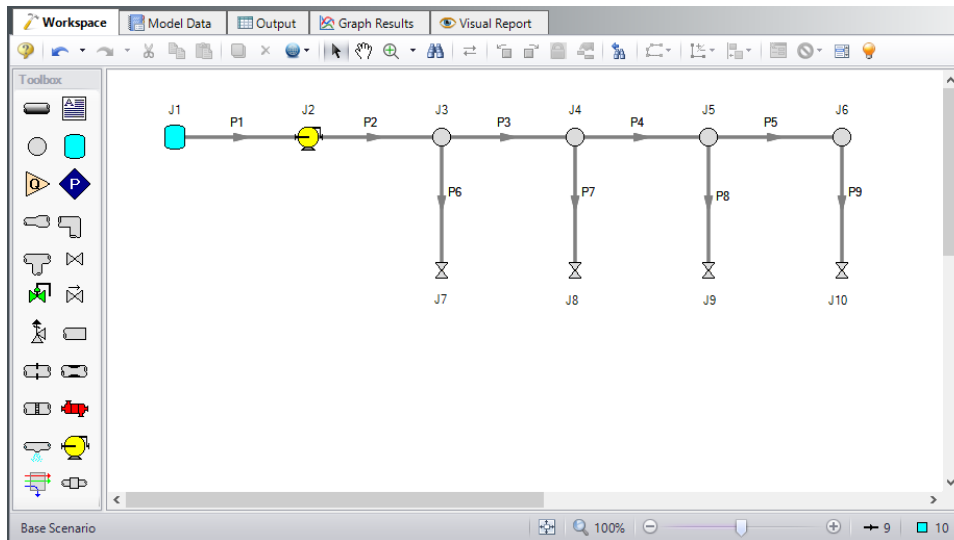
---

## Step 2. Define the Pipes and Junctions Group

---

Open the **US - Compressed Air System.aro** example file listed above, which is located in the Examples folder in the AFT Arrow application folder. Save the file to a different folder.

Once the model is opened and saved to a new location, load the Base Scenario. The workspace should look like Figure 1



**Figure 1: Model Layout for Compressed Air System - Cost Example**

---

## Step 3. Define the Steady Solution Control Group

---

There is an option available in Solution Control when using one of the marching methods that can sometimes result in an overall reduction in solution runtime. This option is to first solve the system using the Lumped Adiabatic method and then use these results as a starting point for the marching solution. Since the Lumped Adiabatic solution can typically be obtained much faster, this can provide an overall reduction in runtime for the marching method.

The Steady Solution Control Group is already defined using the default inputs. This means that no user input is required to run the model. However, the Lumped Adiabatic initialization will be used for this model.

To activate this option, do the following:

1. Open Analysis Setup
2. Navigate to the Solution Method panel in the Steady Solution Control group
3. Click the box to turn on First Use Lumped Adiabatic Method To Obtain Initial Stating Point For Marching Solution

The Steady Solution Control Group is now defined for this model.

---

#### **Step 4. Define the Cost Settings Group**

---

To use the cost calculation feature, navigate to the Cost Settings panel in Analysis Setup.

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.

Energy cost data can be specified through an energy cost library, or by directly entering an energy cost in the Cost Settings window. Using an energy cost library allows you to specify multiple energy costs, such as energy costs at both peak and off-peak rates.

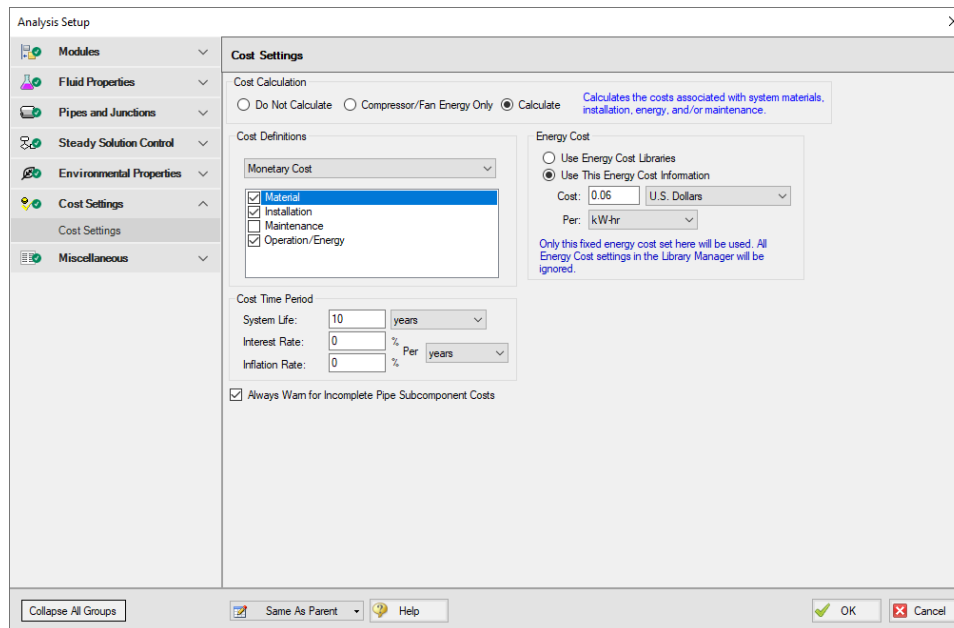
For this example, in the Cost Calculation section at the top, select Calculate. Next, in the Energy Cost section, enter an energy cost of 0.06 U.S. Dollars Per kW-hr, as shown in Figure 2.

The Cost Definitions section allows you to specify the type of costs to be determined. You may select from actual monetary costs, including Material, Installation, Maintenance, and Operation/Energy costs.

For this example, include Material, Installation, and Operation/Energy costs by selecting them from the list of available Monetary Costs, 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 window by clicking OK.



**Figure 2: The cost calculations are enabled on the Cost Settings window**

### Step 5. Engineering and Cost Libraries

The actual material and installation costs that the cost calculation feature uses 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 system component is accessed from a cost library. Cost library items are based on **corresponding items** in an engineering library. (The engineering libraries also include 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**. Moreover, that library must be *connected*.
- There can be multiple cost libraries associated with and connected to an engineering library. This makes it easier to manage costs of items.

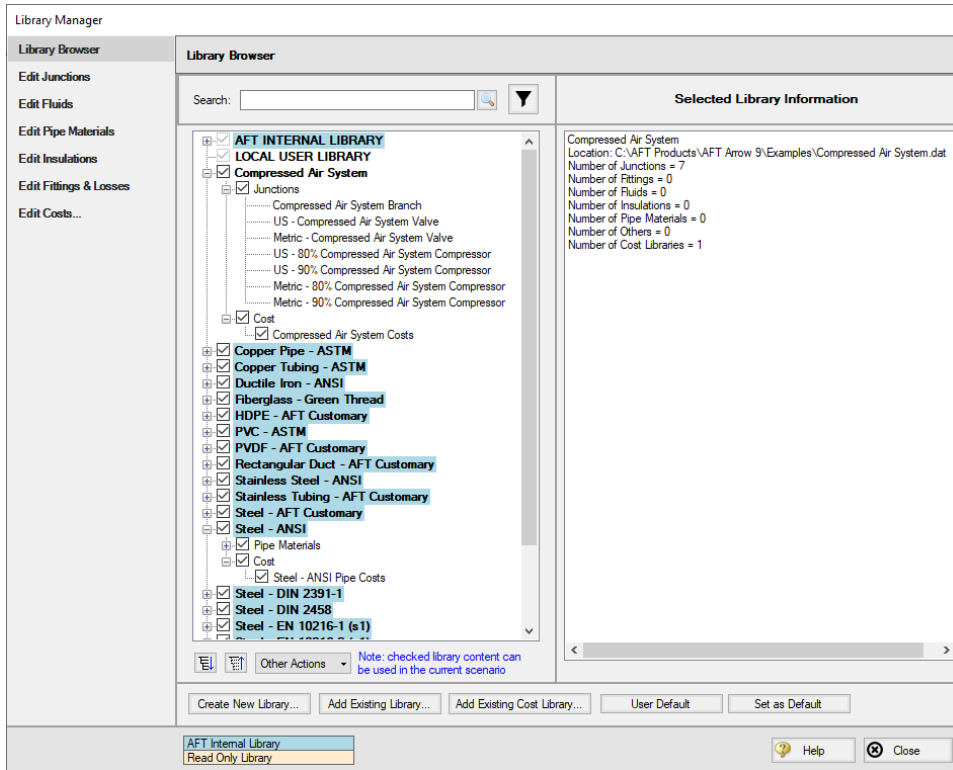
#### **Add the Engineering and Cost Libraries**

Select Library Manager from the Library menu. Add the **Compressed Air System.dat** engineering library file to the list of available libraries by clicking Add Existing Library at the bottom. Browse to the engineering library file in the AFT Arrow 9 Examples folder. The engineering library should now appear in the list of available libraries.

Now add the cost libraries by clicking Add Existing Cost, and browsing to the **Compressed Air System Costs.cst** and the **Steel - ANSI Pipe Costs.cst** cost library files.

All the added libraries should be connected automatically. If not connected, connect the newly added engineering libraries by selecting them in the list of available libraries, and clicking the Add to Connections button.

The Library Manager should appear as shown in Figure 3.



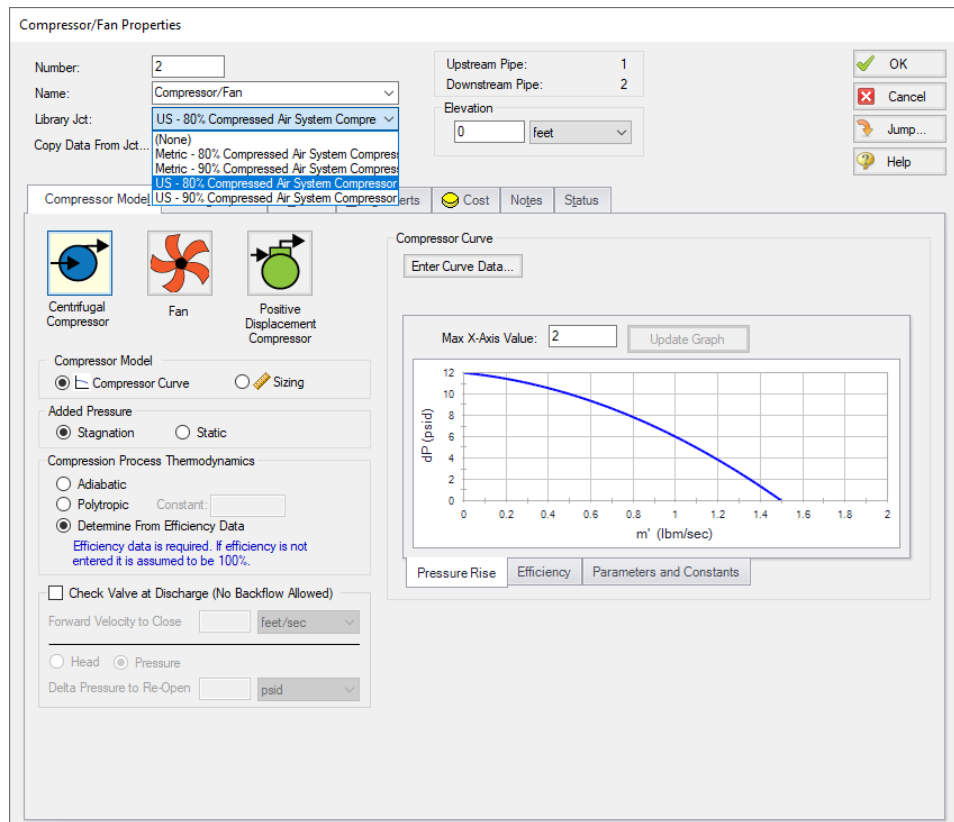
**Figure 3: The Library Manager window displays all of the available and connected libraries**

The engineering libraries associated with these two cost libraries are the Steel - ANSI pipe materials library, and an external library called Compressed Air System. 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 tank junction must be modified to use the component data contained in the engineering library. Do this by opening the Properties window for each junction, and selecting the component name displayed in the Library Jct list, as shown in Figure 4 for the Compressor/Fan junction.

Note that there are two compressor junctions in the library. One uses an efficiency of 80% for the Hot Case scenario, and the other uses an efficiency of 90% for the Cool Case scenario. For this case, select the US 80% efficient compressor.

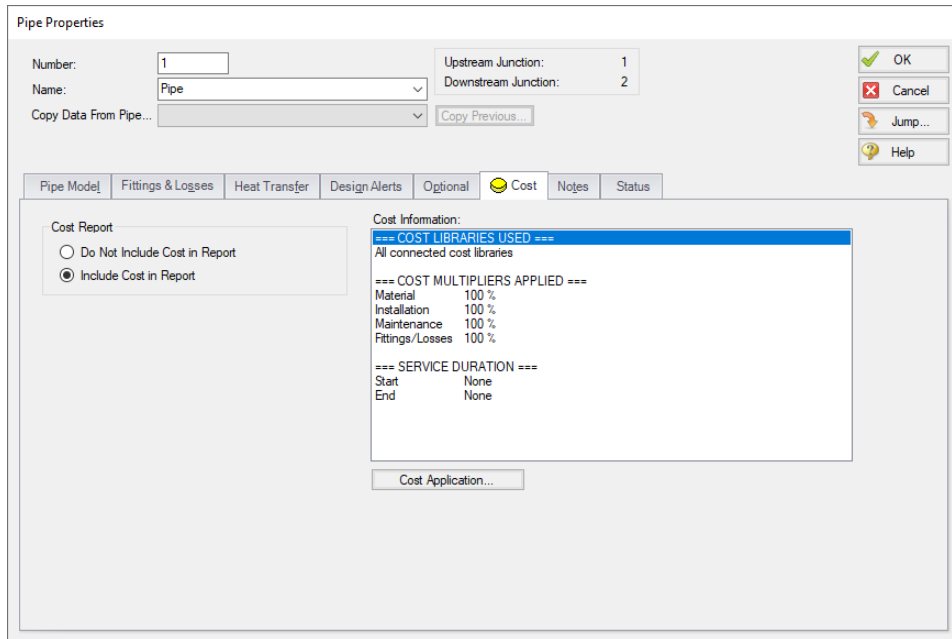


**Figure 4:** The components to be included in the cost analysis must be associated with an engineering library

## Step 6. Include Cost in 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 choosing Include in Cost Report on the Cost tab in the Properties window 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 tabs on the pipe and junction Properties windows are used to include the objects in the Cost Report**

Once all the necessary Pipes and Junctions have been set to be included in the Cost Report, load the Hot Case 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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

## Step 8. Examine the Output

The Cost Report is displayed in the General 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, Operation, 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.



## Compressed Air System - Cost

<span>⤴</span> General   Warnings   Design Alerts   <b>Cost Report</b>   Compressor/Fan Summary   Valve Summary   Tank Summary						
Table Units: U.S. Dollars	Material	Installation	Non-Recurring Sub Total	Operation/Energy	TOTAL	
<b>TOTAL OF ALL MODEL COSTS</b>						<b>97,943</b>
<b>Total of All Shown Costs</b>	<b>10,124</b>	<b>4,463</b>	<b>14,587</b>	<b>83,356</b>	<b>97,943</b>	
<b>Pipe Subtotal</b>	<b>824</b>	<b>863</b>	<b>1,687</b>	<b>0</b>	<b>1,687</b>	
P1	7	7	14	0	14	
P2	170	169	339	0	339	
P3	170	169	339	0	339	
P4	170	169	339	0	339	
P5	170	169	339	0	339	
P6	34	45	80	0	80	
P7	34	45	80	0	80	
P8	34	45	80	0	80	
P9	34	45	80	0	80	
<b>Branch Subtotal</b>	<b>600</b>	<b>900</b>	<b>1,500</b>	<b>0</b>	<b>1,500</b>	
J3	150	225	375	0	375	
J4	150	225	375	0	375	
J5	150	225	375	0	375	
J6	150	225	375	0	375	
<b>Compressor/Fan Subtotal</b>	<b>1,500</b>	<b>500</b>	<b>2,000</b>	<b>83,356</b>	<b>85,356</b>	
J2	1,500	500	2,000	83,356	85,356	
<b>Valve Subtotal</b>	<b>7,200</b>	<b>2,200</b>	<b>9,400</b>	<b>0</b>	<b>9,400</b>	
J7	1,800	550	2,350	0	2,350	
J8	1,800	550	2,350	0	2,350	
J9	1,800	550	2,350	0	2,350	
J10	1,800	550	2,350	0	2,350	

*Figure 6: The Cost Report in the General Output section shows the results of the cost analysis*

### Step 9. Set up and Run the Cold Case Scenario

Load the Cold Case scenario. Repeat Step 4 through Step 8 but using the US - 90% Compressed Air System Compressor for J2 instead. The results from the Cold Case scenario are shown in Figure 7.

## Compressed Air System - Cost

<span>General</span>   <span>Warnings</span>   <span>Design Alerts</span>   <span>Cost Report</span>   <span>Compressor/Fan Summary</span>   <span>Valve Summary</span>   <span>Tank Summary</span>						
Table Units: U.S. Dollars	Material	Installation	Non-Recurring Sub Total	Operation/Energy	TOTAL	
<b>TOTAL OF ALL MODEL COSTS</b>						<b>76,091</b>
<b>Total of All Shown Costs</b>	<b>10,124</b>	<b>4,463</b>	<b>14,587</b>	<b>61,503</b>	<b>76,091</b>	
<b>Pipe Subtotal</b>	<b>824</b>	<b>863</b>	<b>1,687</b>	<b>0</b>	<b>1,687</b>	
P1	7	7	14	0	14	
P2	170	169	339	0	339	
P3	170	169	339	0	339	
P4	170	169	339	0	339	
P5	170	169	339	0	339	
P6	34	45	80	0	80	
P7	34	45	80	0	80	
P8	34	45	80	0	80	
P9	34	45	80	0	80	
<b>Branch Subtotal</b>	<b>600</b>	<b>900</b>	<b>1,500</b>	<b>0</b>	<b>1,500</b>	
J3	150	225	375	0	375	
J4	150	225	375	0	375	
J5	150	225	375	0	375	
J6	150	225	375	0	375	
<b>Compressor/Fan Subtotal</b>	<b>1,500</b>	<b>500</b>	<b>2,000</b>	<b>61,503</b>	<b>63,503</b>	
J2	1,500	500	2,000	61,503	63,503	
<b>Valve Subtotal</b>	<b>7,200</b>	<b>2,200</b>	<b>9,400</b>	<b>0</b>	<b>9,400</b>	
J7	1,800	550	2,350	0	2,350	
J8	1,800	550	2,350	0	2,350	
J9	1,800	550	2,350	0	2,350	
J10	1,800	550	2,350	0	2,350	

**Figure 7: The Cost Report for the Cold Case scenario**

### Cost Analysis Summary

The cost analysis for the Hot Case scenario shows that the total system cost over 10 years for the compressed air system is \$97,943. Of that cost, \$10,124 was for Material costs, \$4,463 was for Installation costs, and \$83,356 was energy cost over the life of the system.

By contrast the cost analysis for the Cold Case scenario shows that the total system cost over 10 years for the compressed air system is \$76,091. Of that cost, \$10,124 was for Material costs, \$4,463 was for Installation costs, and \$61,503 was energy cost over the life of the system.

# Process Steam System - Cost

## Summary

---

This example demonstrates the fundamental concepts of the cost calculation features of AFT Arrow. The example illustrates how the cost calculation features can be used to calculate costs for a given system design.

## 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: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

In addition, it is assumed that the user has worked through the Beginner: [Compressed Air System - Cost](#) example and is familiar with the basics of cost analysis.

## Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Arrow installation:

- ***US - Process Steam System.aro***
- ***US - Process Steam System Costs.aro***
- ***Process Steam System.dat*** - engineering library
- ***Process Steam System Costs.cst*** - completed cost library for ***Process Steam System.dat***
- ***US - Process Steam System Pipe Costs.cst*** - completed cost library for **Steel - ANSI**

## Problem Statement

After designing a process steam plant, it is necessary to calculate the system cost to determine the feasibility of the design. Use the cost calculation features to determine the initial cost of the system. Include material and installation costs for pipes and fittings.

### **Step 1. Start AFT Arrow**

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

---

**Important:** The GSC module and cost calculations can be used in the same run, but we are not going to do this here. Before proceeding, make sure to disable GSC by selecting the radio button for Ignore in the Modules panel in Analysis Setup.

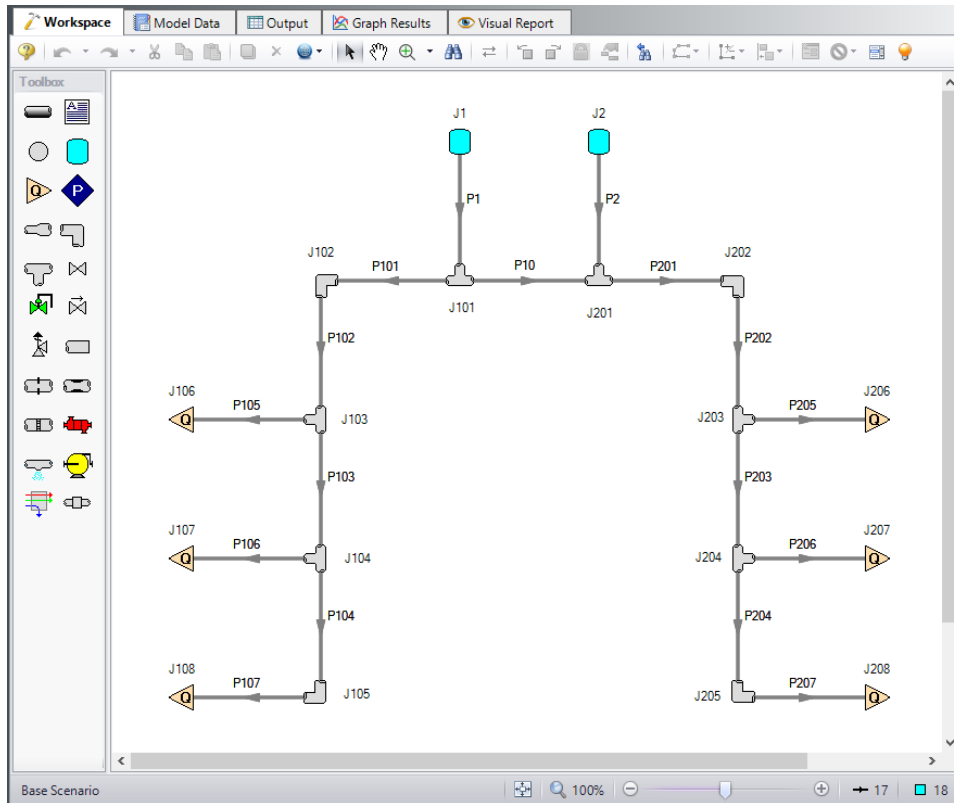
---

### **Step 2. Define the Pipes and Junctions Group**

---

Open the *US - Process Steam System.aro* example file listed above, which is located in the Examples folder in the AFT Arrow application folder. Save the file to a different folder.

Once the model is opened and saved to a new location, load the Base Scenario. This is the only scenario needed for this example, so right-click the Base Scenario and select Delete All Children. The workspace should look like



**Figure 1: Model layout for Process Steam System Cost example**

### Step 3. Define the Cost Settings Group

Open Analysis Setup and go to the Cost Settings panel. Turn on Cost Calculation by selecting the Calculate option at the top.

This example does not have any components for which energy costs can be calculated, so operational/energy costs will not be included in the Cost Report. For this example, leave the energy cost blank.

In the Cost Definitions section, select Material and Installation costs. Additionally, because we are only accounting for material and installation costs, there is no need to enter a system life period for costing purposes.

The Cost Settings panel should now resemble Figure 2

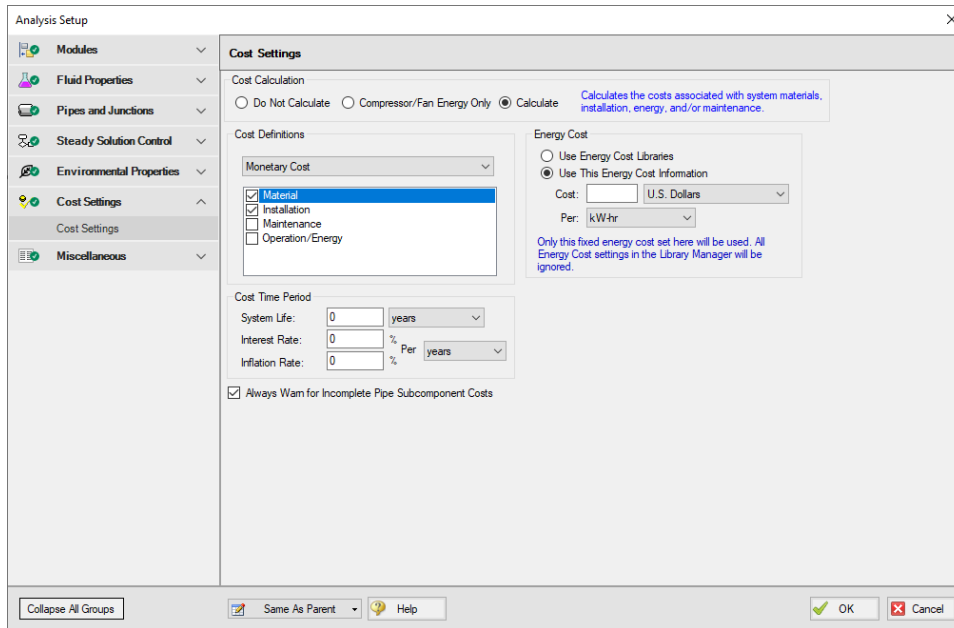


Figure 2: The cost calculations are enabled on the Cost Settings window

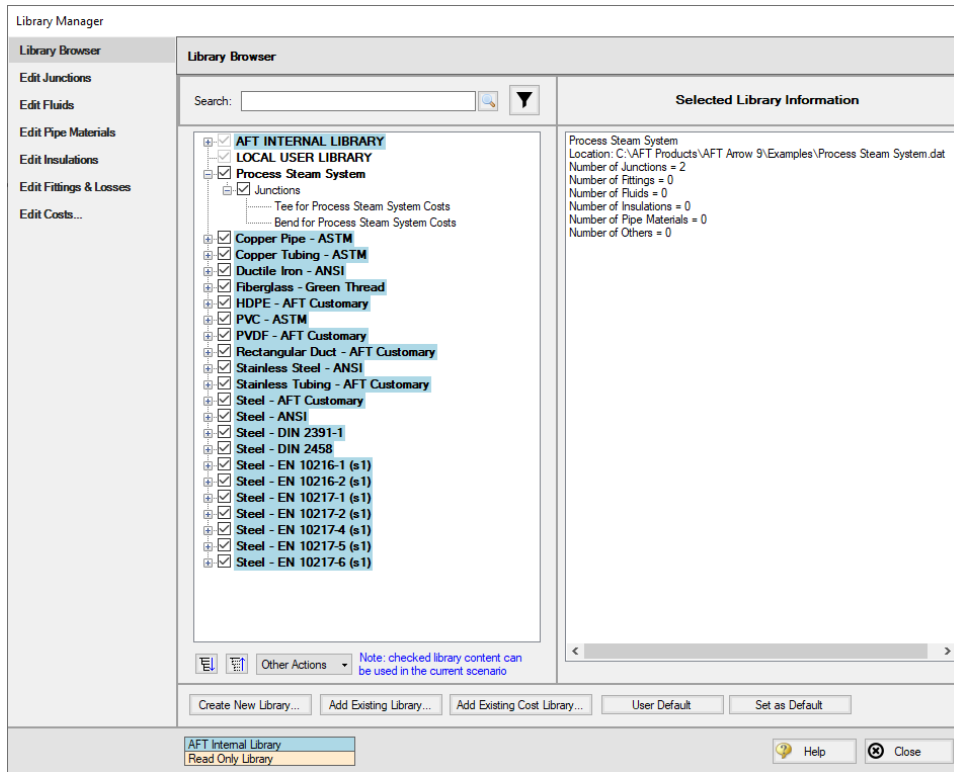
## Step 4. Connect the Engineering Library

For detailed information on using the Library Manager, consult the AFT Arrow Help System.

Select Library Manager from the Library menu. Add the **Process Steam System.dat** engineering library file to the list of available libraries by clicking Add Existing Library at the bottom. Browse to the engineering library file in the AFT Arrow Examples folder. The engineering library should now appear in the list of available libraries.

Make sure the library is connected to the model - indicated by the check mark next to its name.

The Library Manager should appear as shown in Figure 3.



**Figure 3: The engineering library is connected using the Library Manager**

Any pipes and junctions for which costs are to be generated must be associated with an engineering library.

The engineering libraries associated with the junctions in this example are the AFT INTERNAL LIBRARY, and an external library called Process Steam System. The junctions in the model must be associated with one of these two engineering libraries to be able to access the cost data in the cost libraries we will be creating.

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.

The Bend and Tee junctions must be modified to use the component data contained in the external engineering library. Do this by opening the Properties window for each junction, and selecting the component name displayed in the Library Jct list. Make this change for all of the Bend and Tee junctions.

## Step 5. Create the Cost Libraries

Refer to the AFT Arrow Help System for detailed information regarding creating and using cost libraries.

In order to calculate the material and installation cost for the pipes and fittings, these costs must be included. This will be accomplished by creating two new cost libraries. The first library will be associated with the engineering library that contains the pipe material data, and the second library will be associated with the engineering library that contains the fitting data.

## Create a new cost library for the pipe costs

Create a new cost library for the pipe costs by opening the Cost Library window from the Library menu. Create a new cost library by clicking on the New button. When you click on the New button, 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 library, so the pipe cost library will be connected to this library. Select Steel - ANSI from the library list shown in the Select Library window, then click the Select button.

---

**Note:** 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 = US - Process Steam System Pipe Costs
4. Notes = This library is for the Process Steam System Cost Example File

Save the cost library to a file by clicking the Save button. Name the library **US - Process Steam System Pipe Costs.cst**. When prompted, choose to add the new library to the list of available and connected libraries. The information on the General tab should appear as shown in Figure 4. The selected library file information, cost type and monetary units, a description of the cost library, and any notes you have added will be displayed.

---

**Note:** The library file pathnames may be different than those shown in Figure 4.

---

For this example, you will enter Pipe Material costs for the pipes.



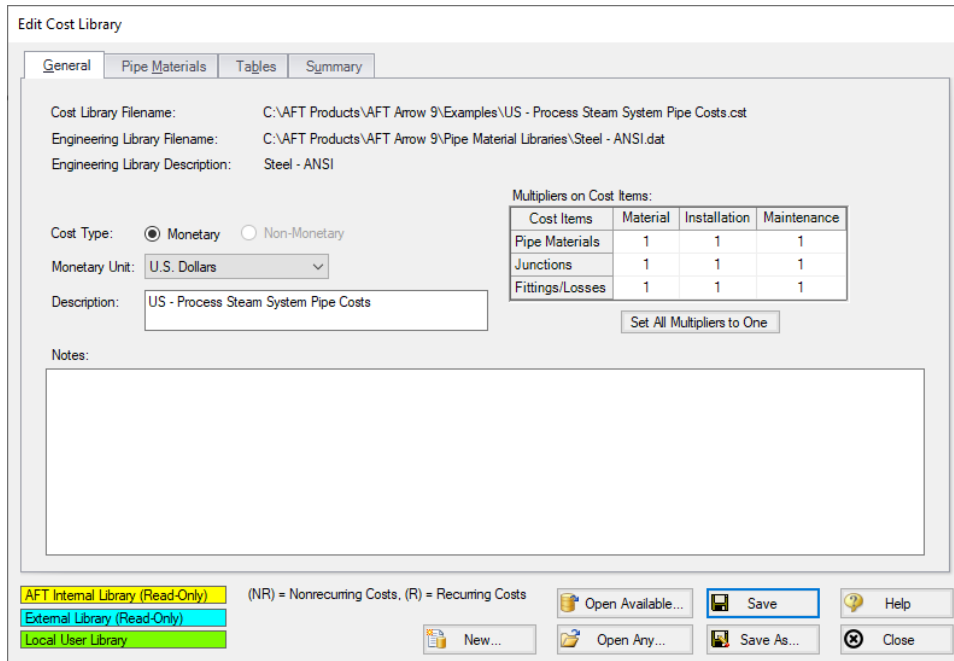


Figure 4: The Edit Cost Library window is used to enter cost data

## Enter the pipe material costs

After the pipe cost library has been created, select the Pipe Materials tab in the Cost Library window. 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 (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.

To enter a new cost, navigate to the material, nominal size and type combination for which you want to enter a cost. Click the New Cost button to create a new cost item in the table below. The new cost item will appear as a new column.

Each pipe size in this example has two non-recurring costs associated with it. The costs to be entered are the material and installation costs. Figure 5 shows the costs entered for 8-inch STD (schedule 40) Steel. Enter the non-recurring pipe costs for the pipe in this example, as shown below.

All pipes are STD (schedule 40) Steel - ANSI:

Cost Type	Material (NR)	Installation (NR)
Size (inches)	Cost (U.S. Dollars)	Cost (U.S. Dollars)
6	8.50	20.00
8	10.50	24.00
12	20.50	51.50
Cost Per	Length	Length
Cost Per Units	feet	feet

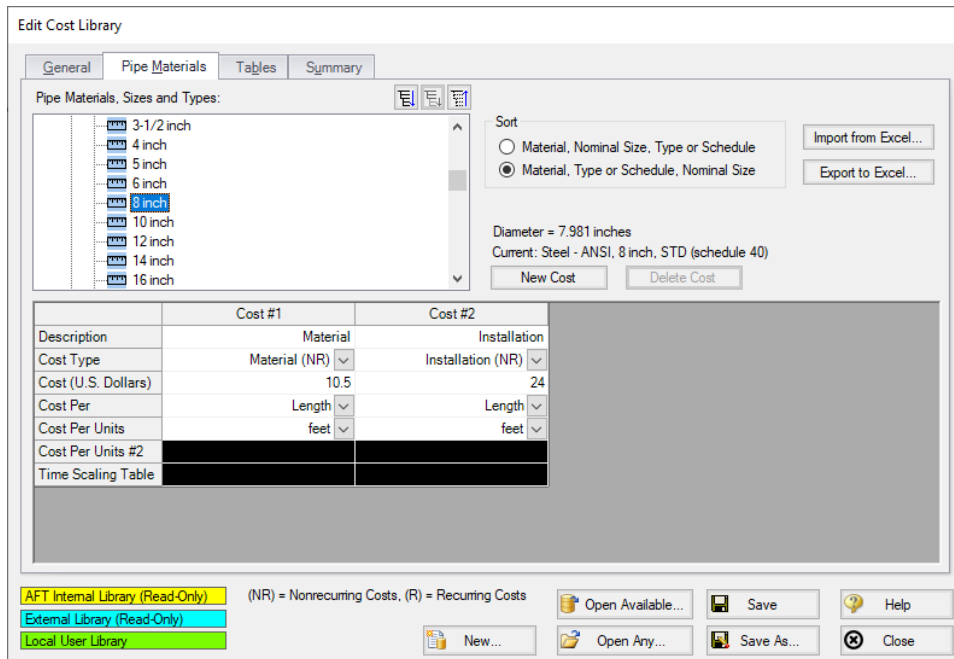


Figure 5: Non-recurring pipe costs, such as material and installation costs, are entered on the Pipe Materials tab

### Create a new cost library for the piping components

To enter cost for a pump, or any other junction type, the junction must first be added to an engineering library. Cost data is then entered in a cost library associated with that engineering library. The piping components (bends and tees) used in this example have been added to an engineering library. Create a new cost library associated with the **Process Steam System.dat** engineering library with the following general information:

1. Cost Type = Monetary
2. Monetary Unit = U.S. Dollars
3. Description = Process Steam System Costs

Save the cost library to a file by clicking the Save button. Name the library Process Steam System Costs.cst. When prompted, choose to add the new library to the list of available and connected libraries.

### Create cost scale tables for the bends

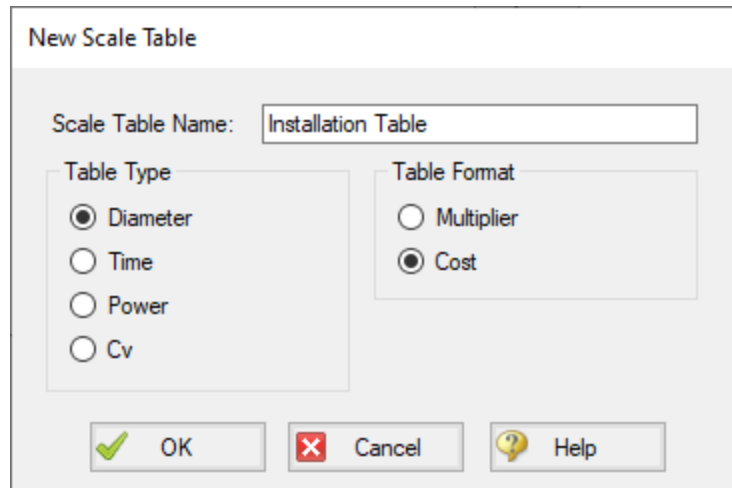
The model contains several 90-degree bend junctions. The cost of these bend fittings can be included in the cost calculation. The costs for these items will be accounted for in scale tables. Scale tables can be used to vary a cost with a parameter such as diameter (i.e., size). Once created, this scale table can be applied to the bend items.

The first table you will create is the scale table for the 90-degree bend installation costs. Select the Tables tab, and click the New Table button.

Enter the following data for the scale table on the New Scale Table window:

1. Name = Installation Table
2. Table Type = Diameter
3. Table Format = Cost

The window should appear as shown in Figure 6. After entering the data, click OK.



**Figure 6: Scale tables are created using the New Scale Table window**

After the table has been created, enter the cost data for the table. Do this by selecting the table name in the Table list, then entering the cost values in the table provided. Enter the following installation cost data into the scale table:

## Process Steam System - Cost

---

<b>Diameter (inches)</b>	<b>Cost (U.S. Dollars)</b>
4	85
6	125
8	185.5
12	298
18	446

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 Arrow 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.

Create another scale table for the material costs using the following data:

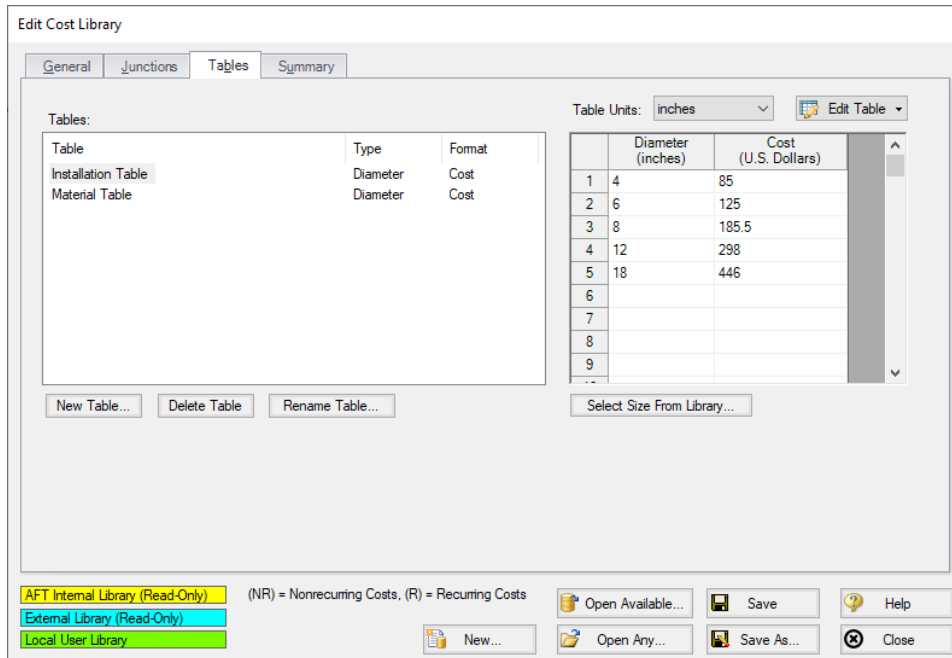
Name = Material Table

Table Type = Diameter

Table Format = Cost

<b>Diameter (inches)</b>	<b>Cost (U.S. Dollars)</b>
4	30
6	45
8	68
12	151.5
18	262.5

After the scale tables have been created, and the cost data entered, the Tables tab should appear as shown in Figure 7.



**Figure 7: The Tables tab on the Cost Library window is used to create cost scale tables**

## Add the costs for the bends

With the cost scale tables for the bends defined, the costs for the bends can be added. Select the Junctions tab. A list of all of the available junctions is displayed in Junction Components list.

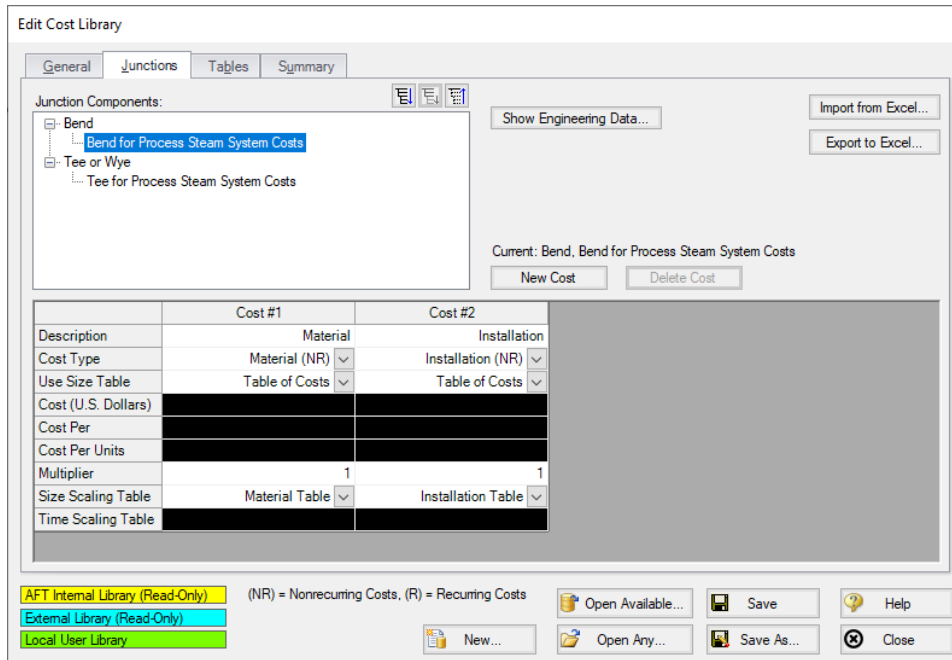
The bends in the model are called Bend for Process Steam System Cost Example. Select this junction and add the material cost by clicking the New Cost button, and entering the following data:

1. Description = Material
2. Cost Type = Material (NR)
3. Use Size Table = Table of Costs
4. Multiplier = 1
5. Size Scaling Table = Material Table

Now add the installation cost by clicking the New Cost button, and entering the following data:

1. Description = Installation
2. Cost Type = Installation (NR)
3. Use Size Table = Table of Costs
4. Multiplier = 1
5. Size Scaling Table = Installation Table

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 Junctions tab should appear as shown in Figure 8.



**Figure 8: The Junctions tab on the Edit Cost Library window is used to specify costs junction items**

### Add the costs for the tees

Add the material cost for the tees by clicking the New Cost button, and entering the following data:

1. Description = Material
2. Cost Type = Material (NR)
3. Use Size Table = (None)
4. Cost (U.S. Dollars) = 150
5. Cost Per = Item

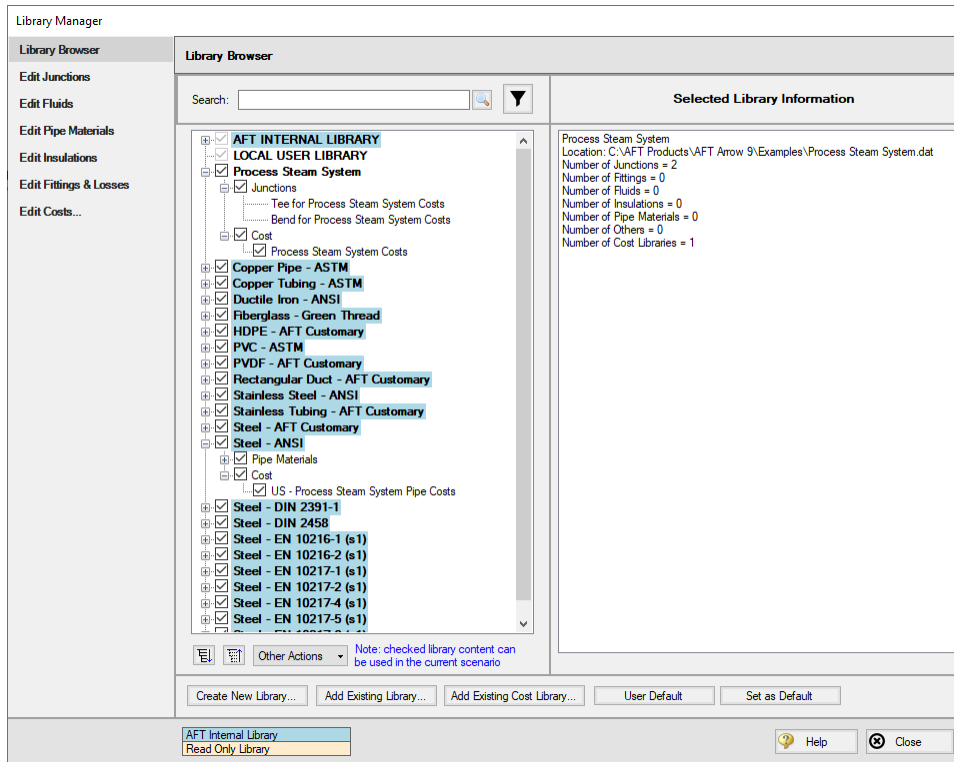
Now add the installation cost for the tees by clicking the New Cost button, and entering the following data:

1. Description = Installation
2. Cost Type = Installation (NR)
3. Use Size Table = (None)
4. Cost (U.S. Dollars) = 225
5. Cost Per = Item

Save the cost library and click the Close button.

## Connecting the cost libraries

The 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 and ). Refer to the AFT Arrow Help system for information on how to connect libraries using the Library Manager.



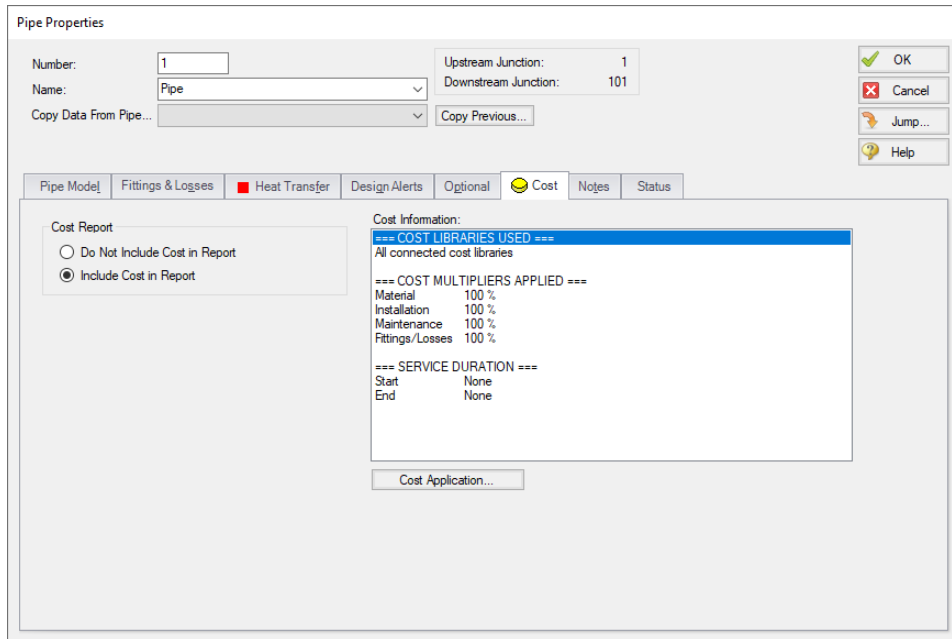
**Figure 9: The Library Manager is used to connect engineering and cost libraries**

## Step 6. Including Items 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 choosing Include in Cost Report on the Cost tab in the Properties window for each pipe and junction to be included, as shown for Pipe 1 in Figure 10.

The Global Edit feature may be used to update this information for all of the pipes and junctions.

For this example, we want to include the costs for all pipes, bends and tees.



**Figure 10:** The Cost tab in the properties window 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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

## Step 8. Examine the Output

The Cost Report is displayed in the General 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 11 shows the Cost Report for this example with the Material, Installation, and Total costs displayed.

The Cost Report shows the total system cost, as well as the individual totals for the material and installation 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.



## Process Steam System - Cost

<span>⤴</span> <span>General</span> <span>Warnings</span> <span>Design Alerts</span> <span>Cost Report</span> <span>Tank Summary</span>				
	Table Units: U.S. Dollars	Material	Installation	TOTAL
<b>TOTAL OF ALL MODEL COSTS</b>				<b>172,611</b>
<b>Total of All Shown Costs</b>		<b>52,184</b>	<b>120,427</b>	<b>172,611</b>
<b>Pipe Subtotal</b>		<b>51,013</b>	<b>118,338</b>	<b>169,350</b>
P1		2,050	5,150	7,200
P2		3,075	7,725	10,800
P10		1,538	3,863	5,400
P101		4,200	9,600	13,800
P102		7,875	18,000	25,875
P103		7,875	18,000	25,875
P104		2,625	6,000	8,625
P105		850	2,000	2,850
P106		425	1,000	1,425
P107		638	1,500	2,138
P201		2,625	6,000	8,625
P202		5,250	12,000	17,250
P203		5,250	12,000	17,250
P204		5,250	12,000	17,250
P205		425	1,000	1,425
P206		638	1,500	2,138
P207		425	1,000	1,425
<b>Bend Subtotal</b>		<b>271</b>	<b>740</b>	<b>1,011</b>
J102		68	185	253
J105		68	185	253
J202		68	185	253
J205		68	185	253
<b>Tee or Wye Subtotal</b>		<b>900</b>	<b>1,350</b>	<b>2,250</b>
J101		150	225	375
J103		150	225	375
J104		150	225	375
J201		150	225	375
J203		150	225	375
J204		150	225	375

*Figure 11: The Cost Report in the General Output section shows the results of the cost analysis*

### Cost analysis summary

The Cost analysis for the Process Steam System shows that the total initial system cost for the process steam system is \$172,611. Of that cost, \$52,184 was for Material costs and \$120,427 was for Installation

costs.



## Module Walk-Through Examples

The examples in the Arrow Module Help System are designed to get new users trained to use the Arrow modules correctly in the shortest amount of time.

If this is your first time using Arrow, please start with the [Walk-Through Examples](#) to learn how to create and run Arrow models. Start with the [Air Heating System](#), because it provides the most complete, basic description on how to set up and run an Arrow 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 Arrow 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.

[Arrow GSC Examples](#)

[Arrow ANS Examples](#)

[Arrow XTS Examples](#)

## Goal Seek and Control (GSC) Examples

The examples below utilize the Goal Seek and Control module.

### Arrow GSC Examples

Example	Complexity	Fluid	Pipes	Compressors	Description
<a href="#">Beginner: Heat Transfer in a Pipe</a>	Beginner	Air	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.
<a href="#">Steam Relief System</a>	Beginner	Steam	6	0	Learn how to use the GSC module to achieve a desired mass flow rate by varying the CdA of a valve.
<a href="#">Natural Gas Burner</a>	Intermediate	Methane	16	0	Learn how to create groups and use the Group Max/Min goal with the GSC module.
<a href="#">Compressed Air System</a>	Beginner	Air	9	1	Learn how to use the GSC module to specify multiple unique variables and goals to achieve a desired system configuration.
<a href="#">Process Steam System</a>	Intermediate	Steam	17	0	Learn how to link GSC variables together to apply multiple variables and goals in a single simulation.

### 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.

**Note:** This example can only be run if you have a license for the GSC module.

---

### Topics Covered

- Using 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 [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

### Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Arrow installation:

- *US - Heat Transfer - GSC.aro*

### Problem Statement

For this example, we will use the GSC module to determine the flow rate required such that air at the outlet of a 200-foot section of pipe, with heat transfer, reaches a temperature of 100 deg. F. The initial air temperature is 300 deg. F at a pressure of 50 psig, the ambient temperature is 70 deg. F, and the ambient air velocity is 15 miles/hr.

### Step 1. Start AFT Arrow

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

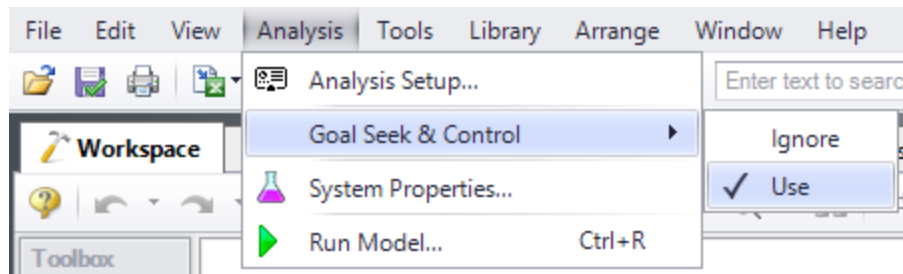
To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

### Step 2. 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 1.



**Figure 1:** Select Use from the Goal Seek & Control menu item on the Analysis menu to instruct AFT Arrow to perform goal seeking when it runs

### Step 3. Define the Fluid Properties Group

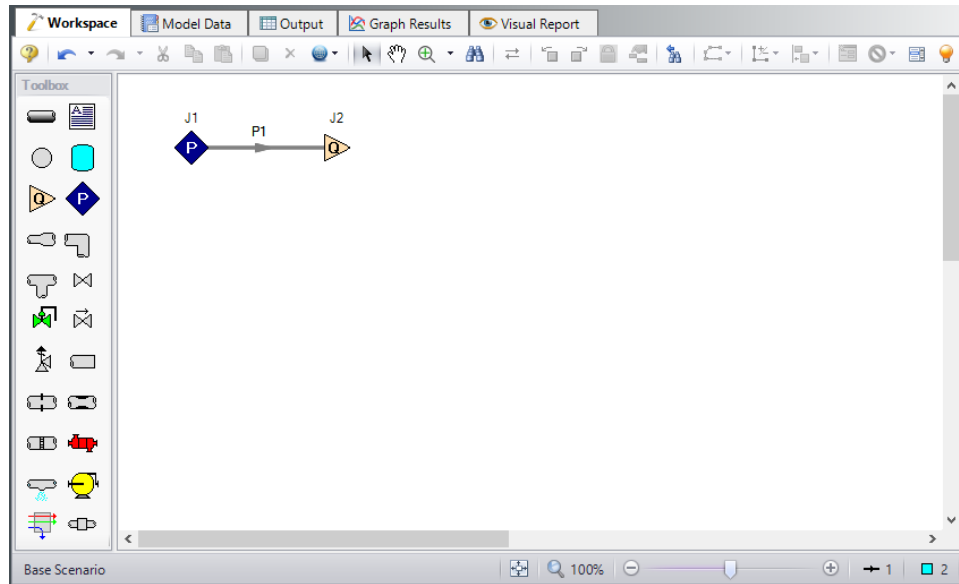
---

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 = Air
    - i. After selecting, click Add to Model
  - c. Equation of State = Redlich-Kwong
  - d. Enthalpy Model = Generalized
  - e. Specific Heat Ratio Source = Library

### 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 2: Model layout for GSC Heat Transfer 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.

## 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. Length = 200 feet
2. Heat Transfer tab
  - a. Heat Transfer Model = Convective Heat Transfer
  - b. Temperature = 70 deg. F
  - c. Fluid Velocity = 15 miles/hr
  - d. Fluid = Air

## Junction Properties

1. J1 Assigned Pressure
  - a. Elevation = 0 feet
  - b. Fluid = Air



- c. Pressure = 50 psig
  - d. Temperature = 300 deg. F
  - e. Pressure Specification = Stagnation
2. J2 Assigned Flow
    - a. Elevation = 0 feet
    - b. Type = Outflow
    - c. Fluid = Air
    - d. Flow Rate = 2 lbm/sec

### **Step 5. Define the Goal Seek and Control Group**

---

The GSC module is controlled through the Goal Seek and Control group in Analysis Setup. The Goal Seek and Control group has three items: Variables, Goals, and Numerical Controls. Use the Variables panel to define the variables. Use the Goals panel to define the goals. and use the Numerical Controls panel to specify the convergence and calculation methods. The default inputs in the Numerical Controls panel will suffice for the vast majority of models using the GSC module.

#### **Variables Panel**

In the GSC module, variables are the parameters that Arrow will modify in order to achieve the specified goals. In order to help maintain unique answers, there should typically be the same number of goals as there are variables.

Open the Variables panel. The Variables panel allows users to create and modify the system variables. The object 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 on the Variables tab.

Click the New Variable button, and enter the following variable data:

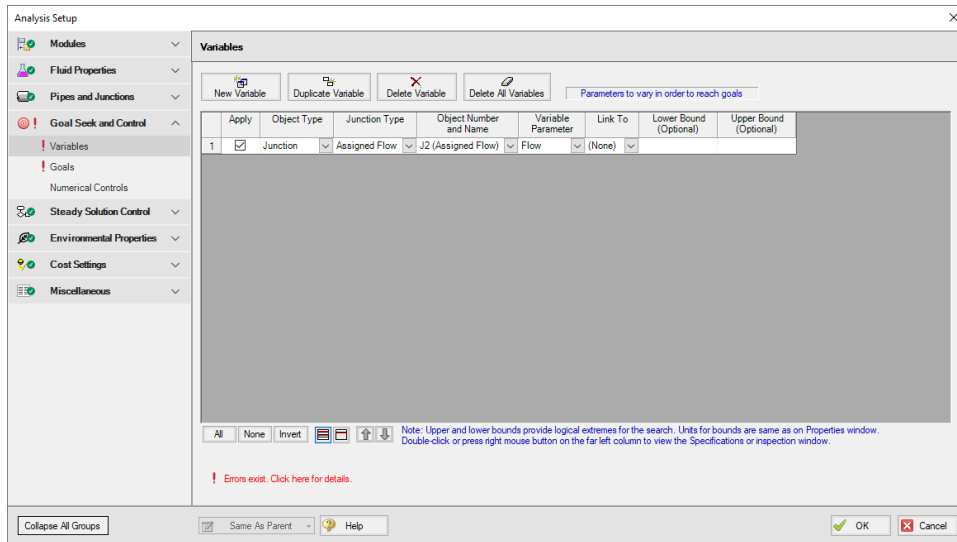
1. Apply = Selected
2. Object Type = Junction
3. Junction Type = Assigned Flow
4. Junction Number and Name = J2 (Assigned Flow)
5. Variable Parameter = Flow
6. Link To = (None)

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. Leave these blank for this example/

After entering the data, the Variable tab should appear as shown in Figure 3:



**Figure 3: The Variables panel is used to specify the variables**

### Goals Panel

Goals are the parameter values set by the user that Arrow adjusts the variables to achieve. Again, there should be the same number of system goals as there are system variables.

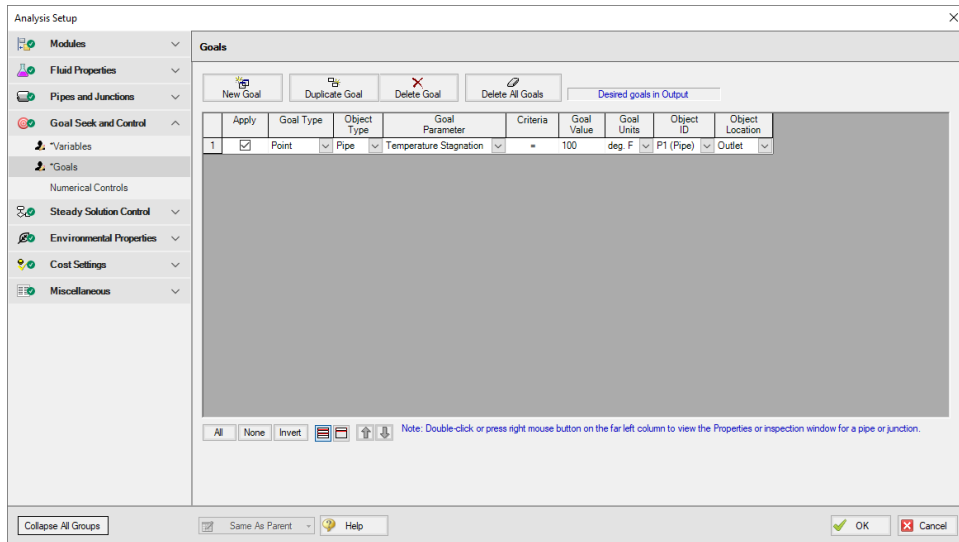
Select the Goals panel in Analysis Setup. The Goals panel allows users to create and modify the system goals. The goal type, object type, and the goal parameter are selected. A criterion for determining if the goal has been met is then specified, along with a value and units for the goal parameter. The user then selects 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).

Click the New Goal button, and enter the following goal data:

1. Apply = Selected
2. Goal Type = Point
3. Object Type = Pipe
4. Goal Parameter = Temperature Stagnation
5. Criteria = =
6. Goal Value = 100
7. Goal Units = deg. F
8. Object ID = P1 (Pipe)
9. Object Location = Outlet

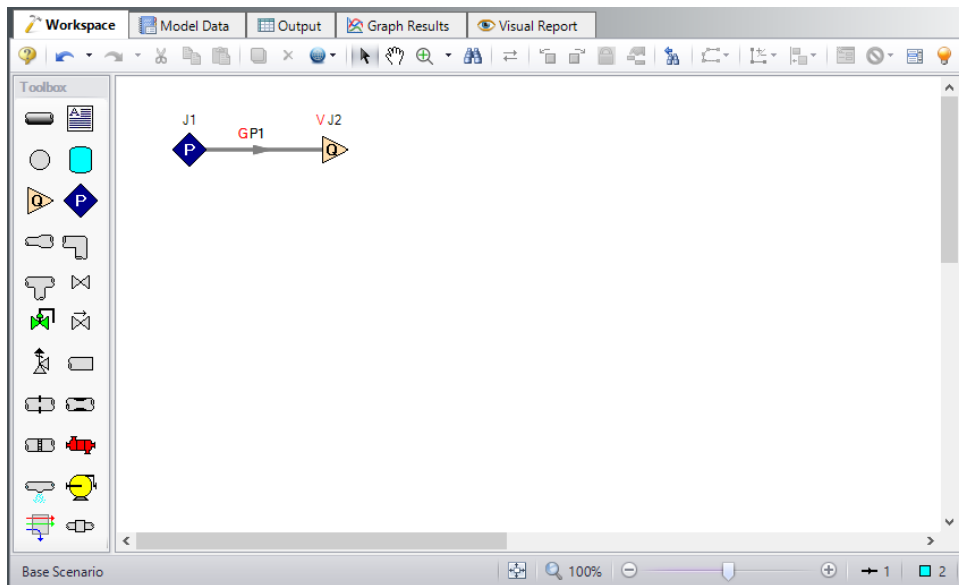
After entering the data, the Goals tab should appear as shown in Figure 4:

## Goal Seek and Control (GSC) Examples



**Figure 4: GSC Manager Goals tab is used to specify the goals**

As variables and goals are added to a model, AFT Arrow 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 5:



**Figure 5: Workspace with Goal and Variable symbols displayed**

## 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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

## Step 7. Examine the Output

The Output window contains all the data that was specified in the Output Control window. Figure 6 shows the Output window.

Execution Time= 2.57 seconds  
 Total Number Of Pressure Iterations= 0  
 Total Number Of Flow Iterations= 170  
 Total Number Of Enthalpy Iterations= 168  
 Number Of Pipes= 1  
 Number Of Junctions= 2  
 Matrix Method= Gaussian Elimination

Pipe	Name	Mass Flow Rate (lbm/sec)	Vel. In (feet/sec)	Vel. Out (feet/sec)	Mach # In	Mach # Out	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. Inlet (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)
1	Pipe	0.4912	24.18	17.82	0.01795	0.01538	0.1361	64.70	64.56	64.68	64.55	200.0	300.0	100.0	300.0	99.97

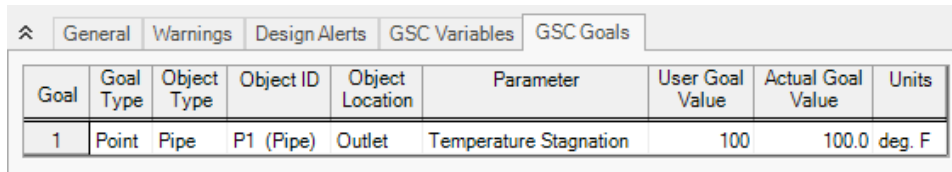
Jct	Name	Mass Flow Rate Thru Jct (lbm/sec)	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. Inlet (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)	dH Stag. In (Btu/lbm)	H Stag. In (Btu/lbm)	H Stag. Out (Btu/lbm)
1	Assigned Pressure	0.4912	0	64.70	64.70	64.68	64.68	0	300.0	300.0	300.00	300.00	0	250.6	250.6
2	Assigned Flow	0.4912	0	64.56	64.56	64.55	64.55	0	100.0	100.0	99.97	99.97	0	202.0	202.0

Figure 6: 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. The GSC Variables tab shows the final value for the variable parameter, as shown in Figure 7. The GSC Goals tab shows the final value achieved for the goal, as shown in Figure 8.

Variable	Object Type	Object	Parameter	Value	Units
1	Assigned Flow	J2	Flow	0.4912	lbm/sec

Figure 7: 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 Stagnation	100	100.0	deg. F

**Figure 8: GSC Goal Results are displayed in the General section of the Output window**

The GSC module analysis determined that a mass flow rate of 0.4912 lbm/sec results in a pipe outlet temperature of 100.0 deg. F. In order to achieve the desired temperature, the assigned flow rate was varied from the initial value of 2 lbm/sec, to the final value of 0.4912 lbm/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.

# Steam Relief System - GSC

## Summary

---

The objective of this example is to demonstrate how goals and variables are used in the GSC module to achieve an objective.

---

**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 [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System 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 Arrow installation:

- *US - Refinery Relief System.aro*
- *US - Refinery Relief System - GSC.aro*

## Problem Statement

For this example, we will use the GSC module to determine the relief valve CdA value required such that the capacity of a steam relief system is a minimum of 30 lbm/sec.

## Step 1. Start AFT Arrow

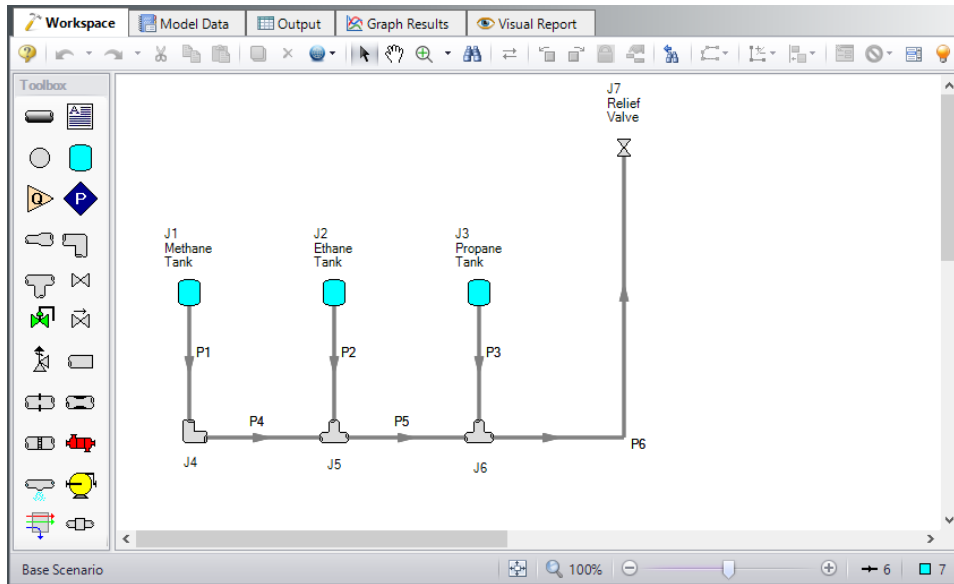
---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

---

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

Open the **US - Refinery Relief System.aro** example file listed above, which is located in the Examples folder in the AFT Arrow application folder. Save the file to a different folder. This is the only scenario needed for this example, so right-click the Base Scenario and select Delete All Children. The workspace should look like Figure 1



**Figure 1: Model layout for GSC Steam Relief System example**

### Step 2. Define the Fluid Properties Group

1. Navigate to the Fluid panel in Analysis Setup
2. Define the Fluid panel with the following inputs
  - a. Fluid Library = AFT Standard
  - b. Fluid = Steam
    - i. After selecting, click Add to Model
  - c. Equation of State = Redlich-Kwong
  - d. Enthalpy Model = Generalized
  - e. Specific Heat Ratio Source = Library

### Step 3. Define the Pipes and Junctions Group

All pipes and junctions should already be defined. However, a few changes need to be made to adapt the model to this example. Make the following changes:

#### Junction Properties

1. J1 Tank
  - a. Name = Steam Process 1
  - b. Fluid = Steam
  - c. Temperature = 400 deg. F
2. J2 Tank
  - a. Name = Steam Process 2
  - b. Fluid = Steam
  - c. Temperature = 400 deg. F
3. J3 Tank
  - a. Name = Steam Process 3
  - b. Fluid = Steam
  - c. Temperature = 400 deg. F

---

#### Step 4. Define the Modules Group

---

Navigate to the Modules panel in Analysis Setup. 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 using the Goal Seek and Control group in Analysis Setup.

#### Variables Panel

Open the Variables panel, click New Variable, and enter the following variable data:

1. Apply = Selected
2. Object Type = Junction
3. Junction Type = Valve
4. Junction Number and Name = J7 (Relief Valve)
5. Variable Parameter = CdA
6. Link To = (None)
7. Lower Bound (Optional) = 0
8. Upper Bound (Optional) = 28

---

**Note:** The units used for the Lower and Upper Bound are the same as what is specified in the junction properties window: inches<sup>2</sup> in this case.

---

#### Goals Panel

Open the Goals panel, click New Goal, and enter the following goal data:

---



1. Apply = Selected
2. Goal Type = Point
3. Object Type = Pipe
4. Goal Parameter = Mass Flow Rate
5. Criteria = =
6. Goal Value = 30
7. Goal Units = lbm/sec
8. Object ID = P6 (Pipe)
9. Object Location = NA

## 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 Arrow solver converges on the answer. Once the solver has converged, 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 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 2. The GSC Goals tab shows the final value achieved for the goal, as shown in Figure 3.

The GSC module analysis determined that a relief valve CdA value of 19.37 inches<sup>2</sup> will result in a relief capacity of 30.00 lbm/sec.

Variable	Object Type	Object	Parameter	Value	Units
1	Valve	J7 (Relief Valve)	CdA	19.37	inches2

**Figure 2: 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	Mass Flow Rate	30	30.00	lbm/sec

**Figure 3: GSC Goal results are displayed in the General section of the Output window**

# Natural Gas Burner - GSC

## Summary

---

The objective of this example is to demonstrate how multiple goals and variables are used in the GSC module to achieve an objective.

---

**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
- Using Group Max/Min Goals
- Understanding Goal Seek and Control output

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System 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 Arrow installation:

- ***US - Natural Gas Burner.aro***
- ***US - Natural Gas Burner - GSC.aro***

## Problem Statement

An underground storage reservoir containing natural gas made up mostly of methane supplies gas to five burners. The gas is at 500 psig, and 50 deg. F. The gas is supplied to each of the burners at 100 psig. To assure that the flow to each of the burners is equal, each burner has a control valve that controls the flow to 5 lbm/sec. The pipes in the system are uninsulated STD (schedule 40) Steel - ANSI, with standard roughness. The ambient temperature surrounding the pipes is 75 deg. F, and the pipes have an ambient air velocity of 15 mph.

In order to operate at the best efficiency, the gas should be delivered to the burners at a minimum stagnation temperature of 100 deg. F. In order to achieve this, the gas flows through a heat exchanger, which has a 10 psi pressure drop at 25 lbm/sec.

How much heat must be added to the heat exchanger to ensure that all of the burners will receive gas at the specified minimum temperature? (Neglect elevation changes)

Use GSC to size the heat exchanger.

## Step 1. Start AFT Arrow

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

Open the **US - Natural Gas Burner.aro** example file listed above, which is located in the Examples folder in the AFT Arrow application folder. Save the file to a different folder. This is the only scenario needed for this example, so right-click the Base Scenario and select Delete All Children. The workspace should look like

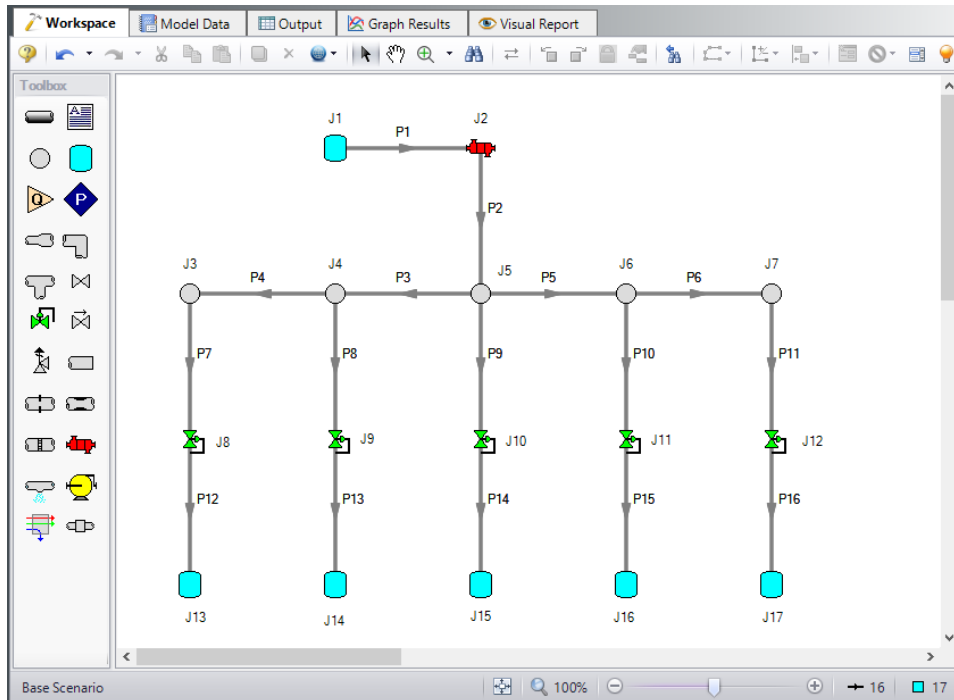


Figure 1: Model layout for GSC Natural Gas Burner example

## Step 2. Define the Steady Solution Control Group

There is an option available in Solution Control when using one of the marching methods that can sometimes result in an overall reduction in solution runtime. This option is to first solve the system using the

Lumped Adiabatic method and then use these results as a starting point for the marching solution. Since the Lumped Adiabatic solution can typically be obtained much faster, this can provide an overall reduction in runtime for the marching method.

The Steady Solution Control Group is already defined using the default inputs. This means that no user input is required to run the model. However, the Lumped Adiabatic initialization will be used for this model.

To activate this option, do the following:

1. Open Analysis Setup
2. Navigate to the Solution Method panel in the Steady Solution Control group
3. Click the box to turn on First Use Lumped Adiabatic Method To Obtain Initial Stating Point For Marching Solution

The Steady Solution Control Group is now defined for this model.

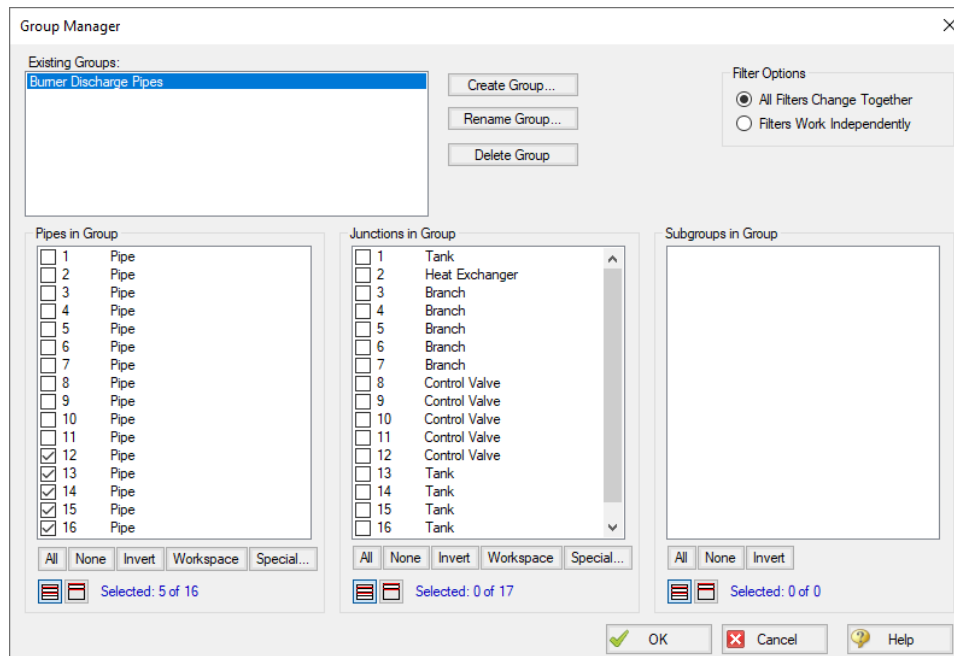
---

### **Step 3. Create a Group**

---

In order to use the Group Max/Min goal type, the junctions or pipes included in the goal must be added to a group. To create a group, select the pipes and/or junctions for the desired group, then select the Groups | Create option from the Edit menu. In this case, only the pipes that are directly connected to the burners will be selected (i.e., Pipes P12-P16).

When prompted, give the new group the name *Burner Discharge Pipes*, and click OK. After naming the new group, the Group Manager will be displayed. Select the new group name from the list, and confirm all of the burner discharge pipes from the pipe list are selected, as shown in Figure 2. Click OK to return to the Workspace.



**Figure 2: Groups are displayed in the Group Manager window**

#### Step 4. Define the Modules Group

Navigate to the Modules panel in Analysis Setup. 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 using the Goal Seek and Control group in Analysis Setup.

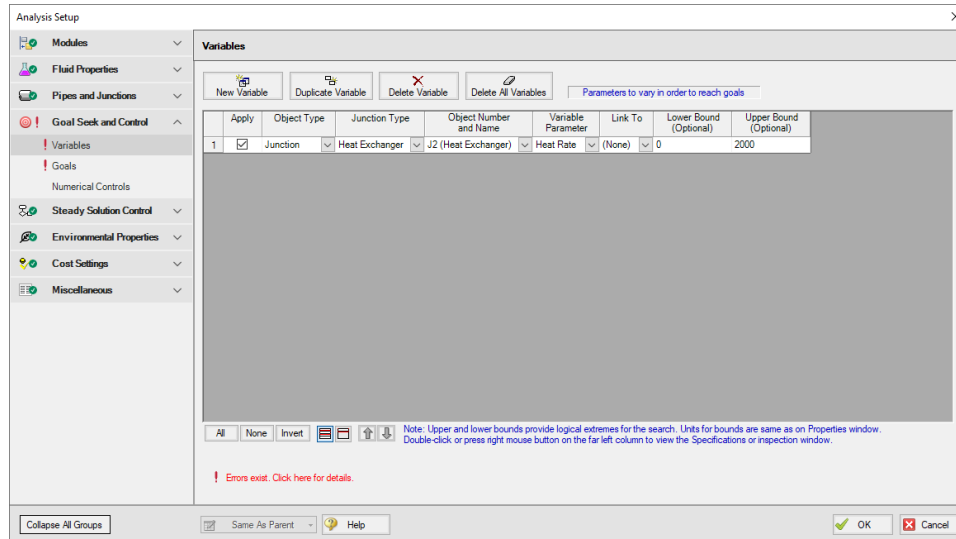
#### Variables Panel

Open the Variables panel, click New Variable, and enter the following variable data (Figure 3):

1. Apply = Selected
2. Object Type = Junction
3. Junction Type = Heat Exchanger
4. Junction Number and Name = J2 (Heat Exchanger)
5. Variable Parameter = Heat Rate
6. Link To = (None)

7. Lower Bound (Optional) = 0
8. Upper Bound (Optional) = 2000

**Note:** The units used for the Lower and Upper Bound are the same as what is specified in the junction properties window: Btu/s in this case.



**Figure 3: Variables are set on the Variables panel in Analysis Setup**

## Goals Panel

For this example, we will be using a Group Max/Min goal. This type of goal allows a single goal to be applied to a group of objects. Arrow 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 our example, a Group Max/Min goal will be applied to ensure the minimum gas discharge temperature out of all of the burner discharge pipes is at least 100 deg. F.

Open the Goals panel, click New Goal, and enter the following goal data:

1. Apply = Selected
2. Goal Type = Group
3. Object Type = Group Max/Min
4. Goal Parameter = Temperature Stagnation
5. Criteria = >=
6. Goal Value = 100
7. Goal Units = deg. F
8. Object ID = Burner Discharge Pipes
9. Object Location = Outlet

## 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 Arrow solver converges on the answer. Once the solver has converged, 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 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 4. The GSC Goals tab shows the final value achieved for the goal, as shown in Figure 5.

The GSC module analysis determined the required heat rate to ensure a minimum gas delivery temperature of 100 deg. F is 998.8 Btu/s. This is the same result that was obtained in the Natural Gas Burner example where the iterations to determine the heat rate were performed manually.

Variable	Object Type	Object	Parameter	Value	Units
1	Heat Exchanger	J2	Heat Rate	998.8	Btu/s

**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	Burner Discharge Pipes	N/A	Temperature Stagnation	100	100.0	deg. F	P12 (Pipe)

**Figure 5: GSC Goal Results are displayed in the General section of the Output window**

# Compressed Air System - GSC

## Summary

---

The objective of this example is to demonstrate how multiple goals and variables are used in the GSC module to achieve an objective.

---

**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
- Applying multiple Variables and Goals
- Interpreting Goal Seek and Control Output

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System 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 Arrow installation:

- *US - Compressed Air System.aro*
- *US - Compressed Air System - GSC.aro*

## Problem Statement

For this example, we will use the GSC module to determine the valve K-factors required to achieve a specified flow of 0.12 lbm/sec to all four tools.



---

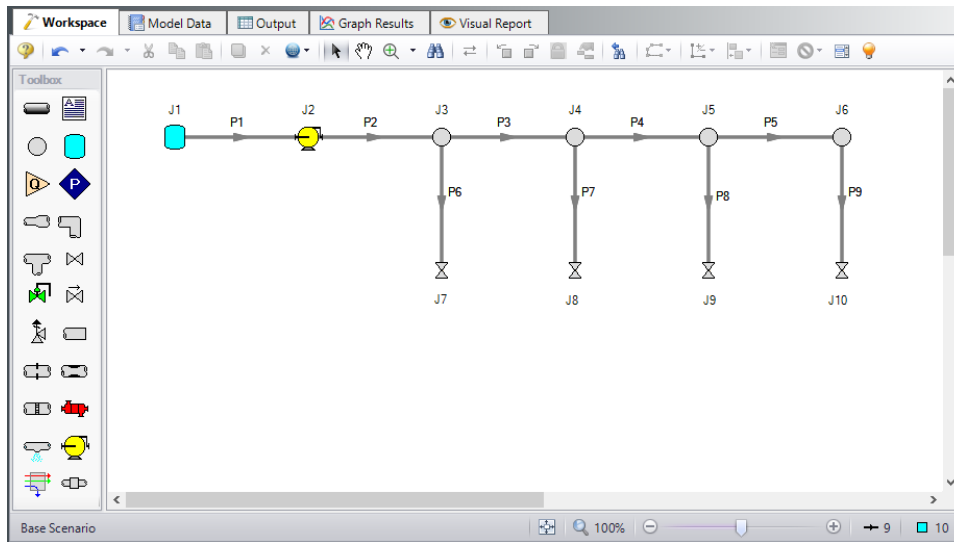
## Step 1. Start AFT Arrow

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

Open the **US - Compressed Air System.aro** example file listed above, which is located in the Examples folder in the AFT Arrow application folder. Save the file to a different folder. This is the only scenario needed for this example, so right-click the Base Scenario and select Delete All Children. The workspace should look like Figure 1



**Figure 1: Model Layout for GSC Compressed Air System Example**

---

## Step 2. Define the Steady Solution Control Group

---

There is an option available in Solution Control when using one of the marching methods that can sometimes result in an overall reduction in solution runtime. This option is to first solve the system using the Lumped Adiabatic method and then use these results as a starting point for the marching solution. Since the Lumped Adiabatic solution can typically be obtained much faster, this can provide an overall reduction in runtime for the marching method.

The Steady Solution Control Group is already defined using the default inputs. This means that no user input is required to run the model. However, the Lumped Adiabatic initialization will be used for this model.

To activate this option, do the following:

1. Open Analysis Setup
2. Navigate to the Solution Method panel in the Steady Solution Control group

3. Click the box to turn on First Use Lumped Adiabatic Method To Obtain Initial Starting Point For Marching Solution

The Steady Solution Control Group is now defined for this model.

---

### Step 3. Define the Pipes and Junctions Group

---

Make the following changes to these junctions:

1. J7-J10 Valves
  1. Subsonic Loss Model = K Factor
  2. K = 10

---

### Step 4. Define the Modules Group

---

Navigate to the Modules panel in Analysis Setup. 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 using the Goal Seek and Control group in Analysis Setup.

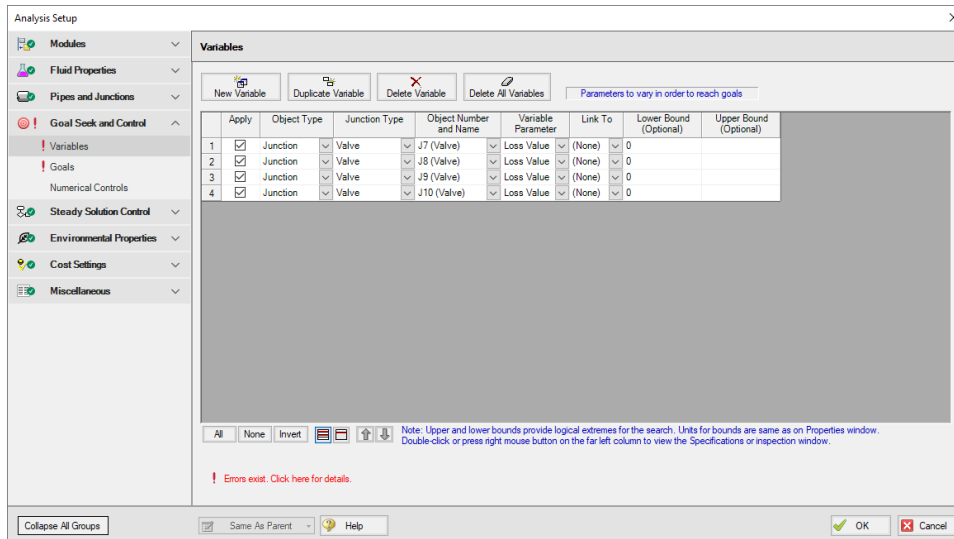
### Variables Panel

Open the Variables panel, click New Variable, and enter the following variable data:

1. J7 Valve
  - a. Apply = Selected
  - b. Object Type = Junction
  - c. Junction Type = Valve
  - d. Junction Number and Name = J7 (Valve)
  - e. Variable Parameter = Loss Value
  - f. Link To = (None)
  - g. Lower Bound = 0
  - h. Upper Bound = Leave blank
2. J8 Valve
  - a. Apply = Selected
  - b. Object Type = Junction
  - c. Junction Type = Valve
  - d. Junction Number and Name = J8 (Valve)

- e. Variable Parameter = Loss Value
  - f. Link To = (None)
  - g. Lower Bound = 0
  - h. Upper Bound = Leave blank
3. J9 Valve
- a. Apply = Selected
  - b. Object Type = Junction
  - c. Junction Type = Valve
  - d. Junction Number and Name = J9 (Valve)
  - e. Variable Parameter = Loss Value
  - f. Link To = (None)
  - g. Lower Bound = 0
  - h. Upper Bound = Leave blank
4. J10 Valve
- a. Apply = Selected
  - b. Object Type = Junction
  - c. Junction Type = Valve
  - d. Junction Number and Name = J10 (Valve)
  - e. Variable Parameter = Loss Value
  - f. Link To = (None)
  - g. Lower Bound = 0
  - h. Upper Bound = Leave blank

After entering the data, the Variables tab should appear as shown in Figure 2.



**Figure 2: Goal Seek and Control group, Variables panel is used to specify the variables**

## Goals Panel

Open the Goals panel, click New Goal, and enter the following goal data:

1. Pipe P6
  - a. Apply = Selected
  - b. Goal Type = Point
  - c. Object Type = Pipe
  - d. Goal Parameter = Mass Flow Rate
  - e. Criteria = =
  - f. Goal Value = 0.12
  - g. Goal Units = lbm/sec
  - h. Object ID = P6 (Pipe)
  - i. Object Location = NA
2. Pipe P7
  - a. Apply = Selected
  - b. Goal Type = Point
  - c. Object Type = Pipe
  - d. Goal Parameter = Mass Flow Rate
  - e. Criteria = =
  - f. Goal Value = 0.12
  - g. Goal Units = lbm/sec
  - h. Object ID = P7 (Pipe)
  - i. Object Location = NA
3. Pipe P8
  - a. Apply = Selected
  - b. Goal Type = Point
  - c. Object Type = Pipe
  - d. Goal Parameter = Mass Flow Rate
  - e. Criteria = =
  - f. Goal Value = 0.12
  - g. Goal Units = lbm/sec
  - h. Object ID = P8 (Pipe)
  - i. Object Location = NA
4. Pipe P9
  - a. Apply = Selected
  - b. Goal Type = Point
  - c. Object Type = Pipe
  - d. Goal Parameter = Mass Flow Rate
  - e. Criteria = =
  - f. Goal Value = 0.12

- g. Goal Units = lbm/sec
- h. Object ID = P9 (Pipe)
- i. Object Location = NA

After entering the data, the Goals tab should appear as shown in Figure 3.

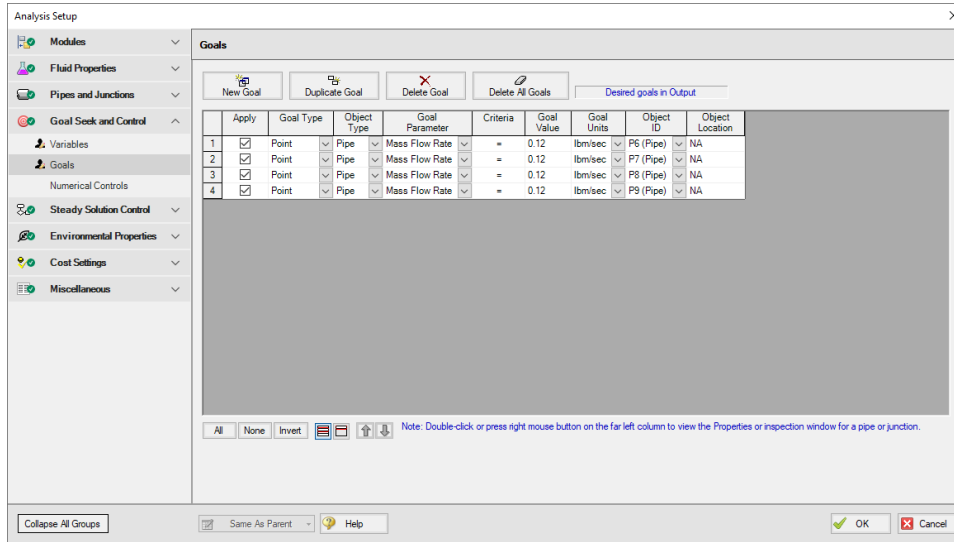


Figure 3: Use the Goals panel is used to specify the goals

## 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 Arrow solver converges on the answer. Once the solver has converged, 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 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 GSC Goals tab shows the final value achieved for the goal, as shown in Figure 5.

The GSC module analysis determined the following valve K-factors would result in the desired flow of 0.12 lbm/sec to each valve:

Object	Value
J7	14.06
J8	11.76

Object	Value
J9	10.71
J10	10.42

Variable	Object Type	Object	Parameter	Value	Units
1	Valve	J7	Loss Value	14.06	N/A
2	Valve	J8	Loss Value	11.76	N/A
3	Valve	J9	Loss Value	10.71	N/A
4	Valve	J10	Loss Value	10.42	N/A

**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
1	Point	Pipe	P6 (Pipe)	N/A	Mass Flow Rate	0.12	0.1202	lbm/sec
2	Point	Pipe	P7 (Pipe)	N/A	Mass Flow Rate	0.12	0.1199	lbm/sec
3	Point	Pipe	P8 (Pipe)	N/A	Mass Flow Rate	0.12	0.1198	lbm/sec
4	Point	Pipe	P9 (Pipe)	N/A	Mass Flow Rate	0.12	0.1199	lbm/sec

**Figure 5: GSC Goal Results are displayed in the General section of the Output window**

# Process Steam System - GSC

## Summary

---

The objective of this example is to demonstrate how multiple goals and variables are used in the GSC module to achieve an objective.

---

**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
- Using Group Max/Min Goals
- Linking Variables
- Interpreting Goal Seek and Control output

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System 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 Arrow installation:

- *US - Process Steam System.aro*
- *US - Process Steam System - GSC.aro*

## Problem Statement

For this example, we will use the GSC module to determine the supply pressure necessary to meet a minimum delivery pressure of 100 psig at each discharge location for the specified flows.

## Step 1. Start AFT Arrow

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

Open the **US - Process Steam System.aio** example file listed above, which is located in the Examples folder in the AFT Arrow application folder. Save the file to a different folder. This is the only scenario needed for this example, so right-click the Base Scenario and select Delete All Children. The workspace should look like Figure 1:

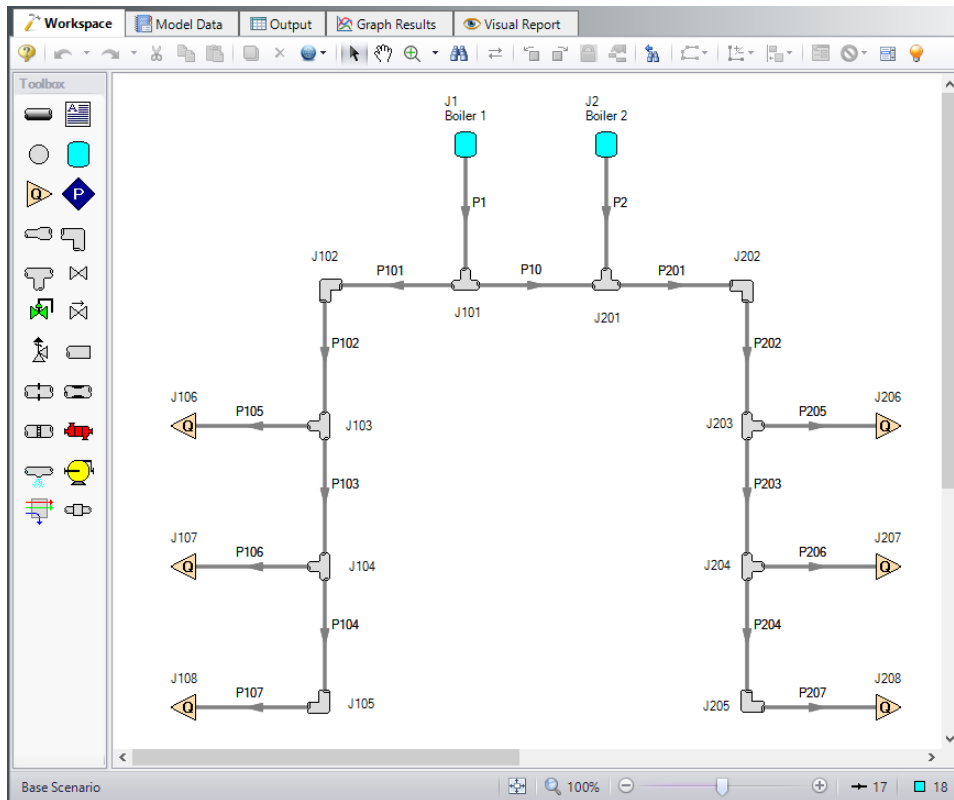
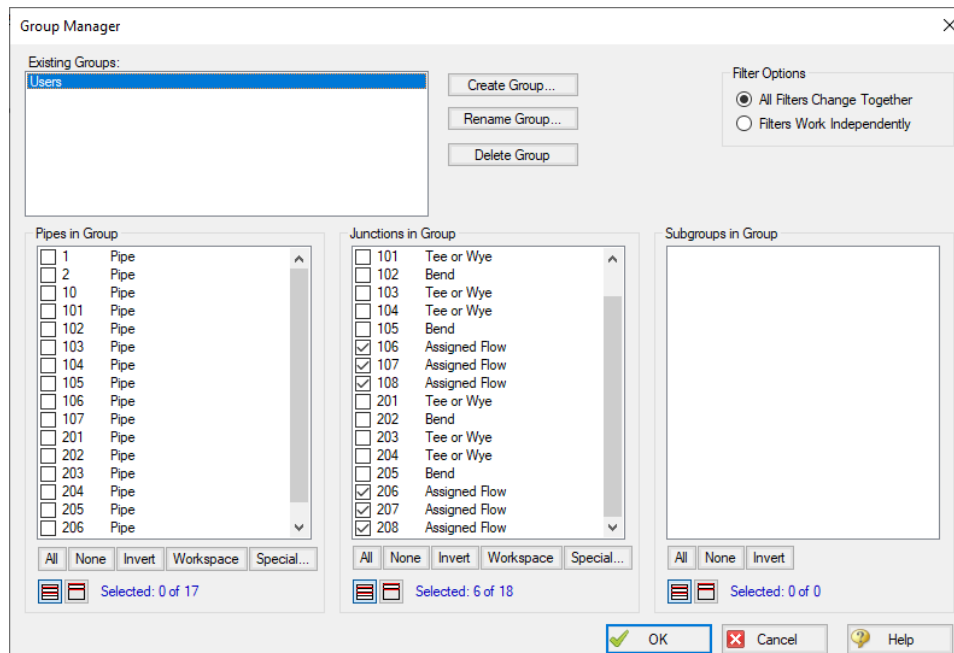


Figure 1: Model layout for GSC Process Steam System example

## Step 2. Create a Group

In order to use the Group Max/Min goal type, the junctions or pipes included in the goal must be added to a group. To create a group, select the Groups | Create option from the Edit menu. When prompted, give the new group the name *Users* and click OK. After naming the new group, the Group Manager will be displayed. Select the new group name from the list, and then select all of the assigned flow junctions from the junction list, as shown in Figure 2. After choosing the assigned flow junctions, click OK on the Group Manager window.





**Figure 2: Groups are created using the Group Manager window**

### Step 3. Define the Modules Group

Navigate to the Modules panel in Analysis Setup. 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 using the Goal Seek and Control group in Analysis Setup.

#### Variables Panel

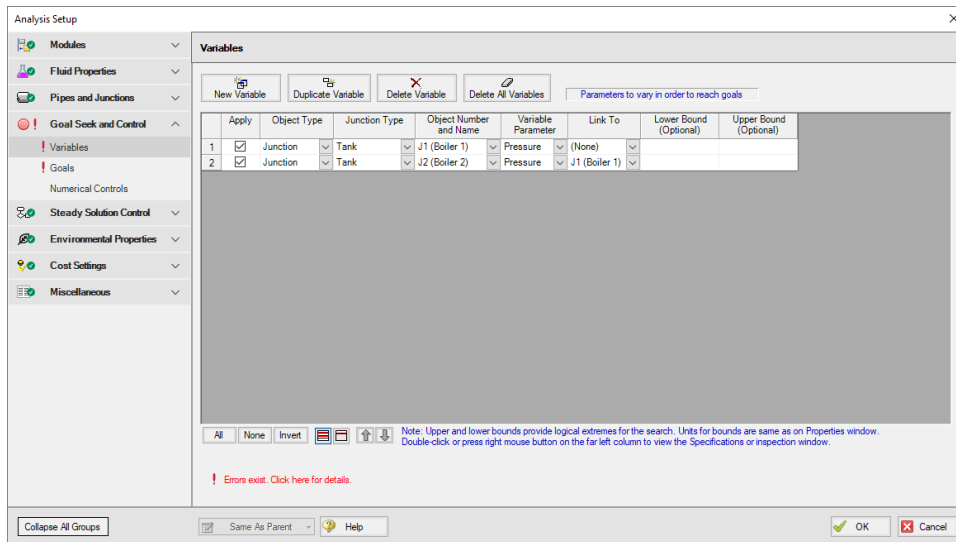
For this example, we want to find the supply pressures that will give us a specific minimum discharge pressure. We will define the delivery pressures as a single Group Max/Min goal. But that gives us only one goal for two variables. To solve this issue, we will use variable linking. Variable linking allows you to tie multiple variables together, forcing them to be the same. To accomplish this, we will add a constraint to the problem that both of the boiler supply pressures must be the same.

To implement variable linking, add both tank junctions pressures as variables. Then link them together by specifying Boiler 1 (J1) in the Link To column of the Boiler 2 (J2) variable. Enter the data as shown below.

Open the Variables panel, click New Variable, and enter the following variable data:

1. J1 Boiler 1
  - a. Apply = Selected
  - b. Object Type = Junction
  - c. Junction Type = Tank
  - d. Junction Number and Name = J1 (Boiler 1)
  - e. Variable Parameter = Pressure
  - f. Link To: (None)
  - g. Lower Bound: Leave blank
  - h. Upper Bound: Leave blank
2. Boiler 2 (J2)
  - a. Apply = Selected
  - b. Object Type = Junction
  - c. Junction Type = Tank
  - d. Junction Number and Name = J2 (Boiler 2)
  - e. Variable Parameter = Pressure
  - f. Link To = J1 (Boiler 1)
  - g. Lower Bound: Leave blank
  - h. Upper Bound: Leave blank

After entering the variable data, the Variables tab in the Goal Seek and Control manager should appear as shown in Figure 3.



**Figure 3: Variable solutions can be linked together using the Link To option**

## Goals Panel

For this example, we will be using a Group Max/Min goal. This type of goal allows a single goal to be applied to a group of objects. Arrow 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 our example, a Group Max/Min goal will be applied to ensure the minimum discharge pressure at all of the assigned flow junctions is at least 100 psig.

Open the Goals panel, click New Goal, and enter the following goal data:

1. Apply = Selected
2. Goal Type = Group
3. Object Type = Group Max/Min
4. Goal Parameter = Pressure Stagnation
5. Criteria = >=
6. Goal Value = 100
7. Goal Units = psig
8. Object ID = Users
9. Object Location = NA

As variables and goals are added to a model, AFT Arrow 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, so you will see only the V displayed on the Workspace for this example.

---

## Step 5. Run the Model

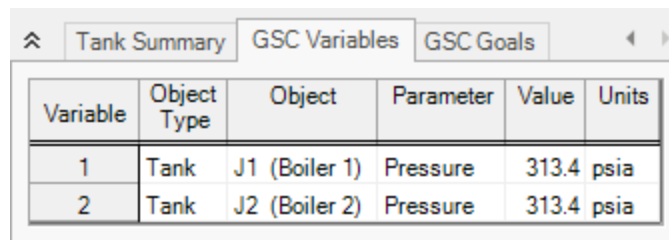
Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

---

## Step 6. Examine the Output

The results of the GSC 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 4. The GSC Goals tab shows the final value achieved for the goal, as shown in .

The GSC module analysis determined the supply pressure required to ensure a minimum delivery pressure of 100 psig at the users is 313.4 psia.



Variable	Object Type	Object	Parameter	Value	Units
1	Tank	J1 (Boiler 1)	Pressure	313.4	psia
2	Tank	J2 (Boiler 2)	Pressure	313.4	psia

*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	Object Used for Group Goal
1	Group	Group Max/Min	Users	N/A	Pressure Stagnation	100	100.0	psig	J108 (Assigned Flow)

**Figure 5: GSC Goal Results are displayed in the General section of the Output window**

## Automatic Network Sizing (ANS) Examples

The examples below utilize the Automated Network Sizing module.

### Arrow ANS Examples

Example	Complexity	Fluid	Pipes	Compressors	Description
<a href="#">Beginner: Three Tank Steam Sys- tem</a>	Beginner	Steam	3	0	Learn the basics of the ANS module such as defining all panels in the Sizing window to achieve a minimum pipe weight objective
<a href="#">Control Valve</a>	Beginner	Steam	2	0	Learn how to use the ANS module and how Common Size Groups work to minimize the pipe weight of a system.
<a href="#">Air Dis- tribution, Creating Libraries</a>	Intermediate	Air	18	1	Learn how to build cost libraries and engineering libraries for use with the ANS module. See how to use Scale Tables for defining compressor costs, how to specify material versus installation costs, and how to define costs for pipes.
<a href="#">Air Dis- tribution</a>	Intermediate	Air	18	1	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.
<a href="#">Air Dis- tribution, Mul- tiple Design Cases</a>	Intermediate	Air	18	1	Learn how to use the ANS module to minimize monetary cost using a Dependent Design Case.
<a href="#">Helium Trans- fer</a>	Beginner	Helium	3	0	Learn how to use the ANS module to minimize the pipe

Example	Complexity	Fluid	Pipes	Compressors	Description
					weight of a system based on a dependent design configuration, called a Dependent Design Case.
<a href="#">Natural Gas Burner</a>	Intermediate	Methane	16	0	Learn how to use the ANS module to minimize the monetary cost using a Dependent Design Case then comparing the results to the same system but with an objective to minimize pipe weight.
<a href="#">Process Steam System</a>	Intermediate	Steam	17	0	Learn how to use the ANS module and how the Search Method impacts results

## Beginner: Three Tank Steam System - ANS

### Summary

The objective of this example is to familiarize the user with the components of 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 pipe weight to minimize cost
- Defining Pipe Design Requirements
- Choosing Candidate Sets

### Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

### Model Files

This example uses the following files, which are installed in the Examples folder as part of the AFT Arrow installation:

- **US - Three Tank System - ANS Initial.aro**
- **US - Three Tank System - ANS Final.aro**

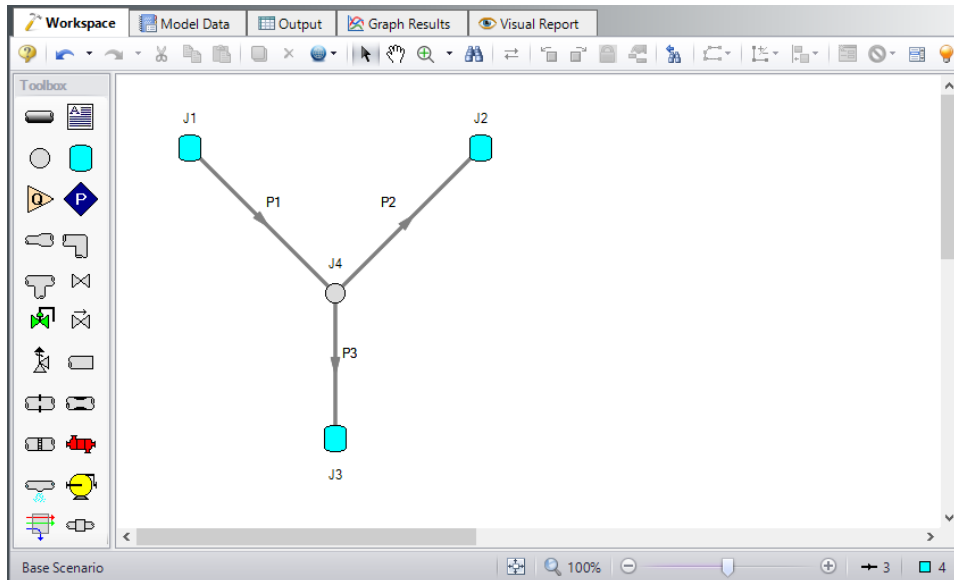
### Step 1. Start AFT Arrow

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

Open the **US - Three Tank System - ANS Initial.aro** example file listed above, which is located in the Examples folder in the AFT Arrow application folder. Save the file to a different folder. This is the only scenario needed for this example, so right-click the Base Scenario and select Delete All Children. The workspace should look like Figure 1:

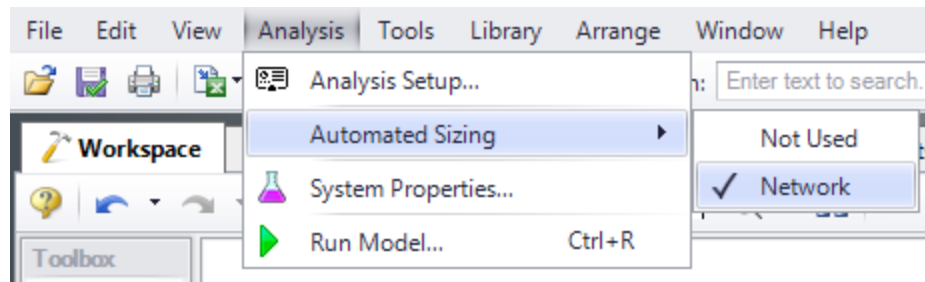


**Figure 1: Layout of Three Tank System model in Workspace**

### Step 2. Define the Modules Group

---

The ANS module is controlled through the Sizing window which is activated and enabled through Analysis Setup. Open Analysis Setup, and navigate to the Modules panel. Check the box next to Activate ANS. The Network option should automatically be selected, making ANS enabled for use. Alternatively, ANS can be enabled from the Analysis menu by going to the Automated Sizing option and selecting Network, as shown in Figure 2.



**Figure 2: Activating Network Sizing from the Analysis menu**

### Step 3. Define the Automatic Sizing Group

---

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 Arrow Help file.

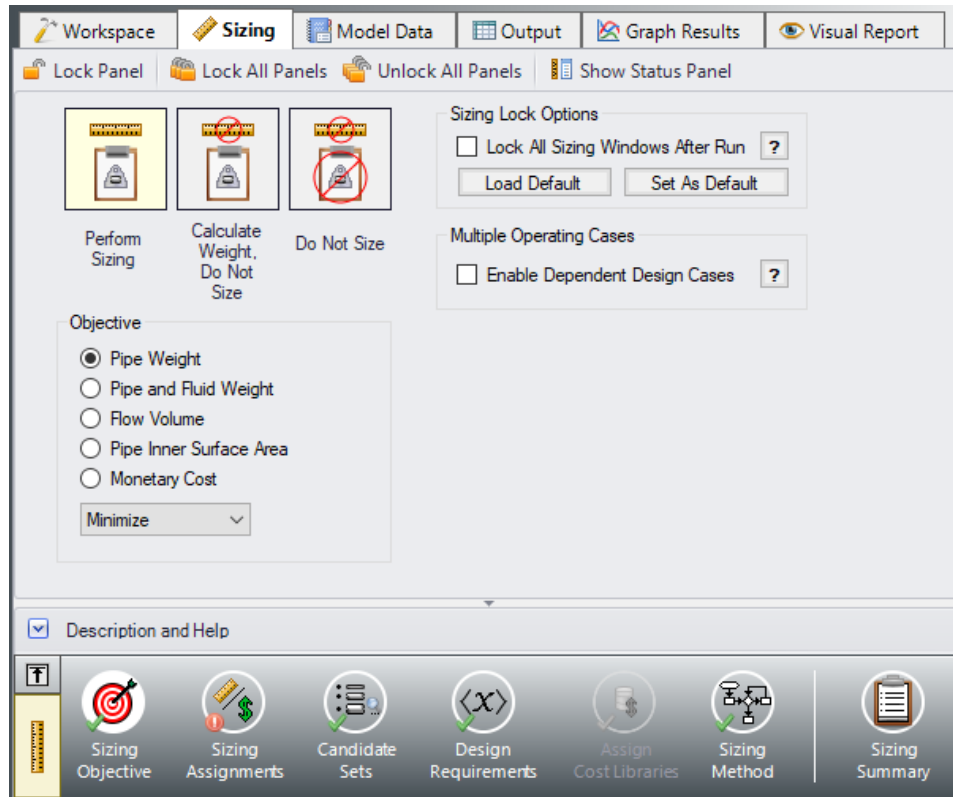
The Sizing window is comprised of multiple panels which can be accessed using the buttons on the Sizing Navigation Panel along the bottom of the window, as shown in Figure 3. The sizing panels can be accessed in any order, though it is easiest to enter the information by navigating the panels from left 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.

Similar to Analysis Setup, 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 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. This setting can be changed on the Sizing Objective panel, as can be seen in Figure 3.





**Figure 3: Sizing Window showing common sizing panel features**

## A. Sizing Objective

The Sizing Objective panel is used to set 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 Arrow without requiring any cost or sizing information. This is useful to preserve the information already entered in the Sizing window while running the model normally.

➤ **Select Perform Sizing** to begin configuring the sizing settings.

---

**Note:** The Sizing Level Status, which shows the type of calculation being performed, 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 next step is to set a Sizing Objective, which can be monetary cost, or a non-monetary cost such as pipe weight or flow volume. It is easier to perform a non-monetary sizing, since all of the geometry and material information necessary to calculate the pipe weight/volume is already contained in the pipe material libraries. A monetary cost objective would require cost information to be obtained and entered into

cost libraries. For many systems, sizing the system using one of the simpler non-monetary objective options provides a sufficient approximation for minimized cost. In this case, we will be performing a minimization for pipe weight.

➤ **For the Objective, choose Pipe Weight**, and select the Minimize option from the drop-down list if not already selected, as shown in Figure 3 above.

The other option available in the Sizing Objective panel is to configure the Sizing Lock Options as mentioned previously. This option will be left as the default.

### 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 compressors/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 this case since the Sizing Objective is Pipe Weight we will discuss all costs in terms of weight.

In the Sizing and Weight 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 weight calculation by choosing Always Include in Weight. If you are instead analyzing the possible replacement of existing pipes, it may be better to size the pipes, but only include the weight if the size changes from that which exists by choosing Include in Weight if Size Changes.

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 be unrealistic.

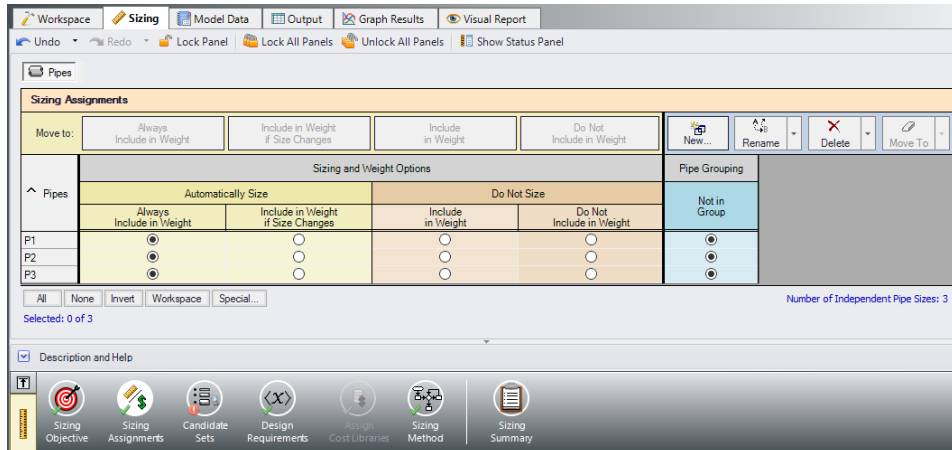
Another reason may be if a certain size is necessary for the design due to certain requirements, in which case the pipe weight can be included without sizing the pipe by choosing Include in Weight.

This model is a new system, so all pipes will be sized and included in the weight calculation.

➤ **Select Always Include in Weight** under Automatically Size for all three pipes (Figure 4).

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 pipe weight. While this value can easily be calculated for the pipes, Arrow does not have enough information to calculate a weight 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 flow volume. If the size of a junction will have a large impact on the system cost, such as a compressor, Monetary Cost should be used as the Objective.



**Figure 4: Sizing Assignments panel for Three Tank 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:

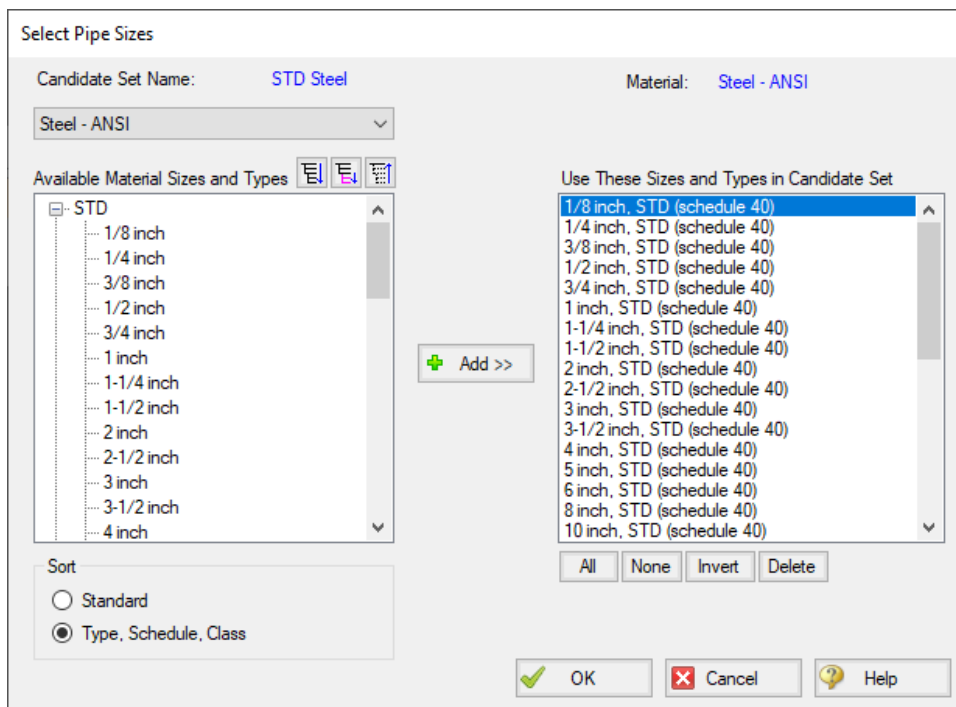
1. Under Define Candidate Sets, click New.
2. Give the set the name *STD Steel* and click OK.
3. From the drop down list choose Steel - ANSI.
4. In the bottom of the Select Pipe Sizes window, click the Sort option Type, Schedule, Class if it is not already selected.

5. In the Available Material Sizes and Types list on the left, select STD so that the schedule name is now highlighted.
6. Click the Add button to add all STD pipe sizes to the list on the right (Figure 5).
7. In the Select Pipe Sizes window, click the OK button.

The STD Steel set will now appear in the Define Candidate Sets section.

We will apply the STD Steel 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 STD Steel - ANSI. At the low end this includes 1/8th 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 STD Steel 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 5: Select Pipe Sizes window for defining a new Candidate Set**

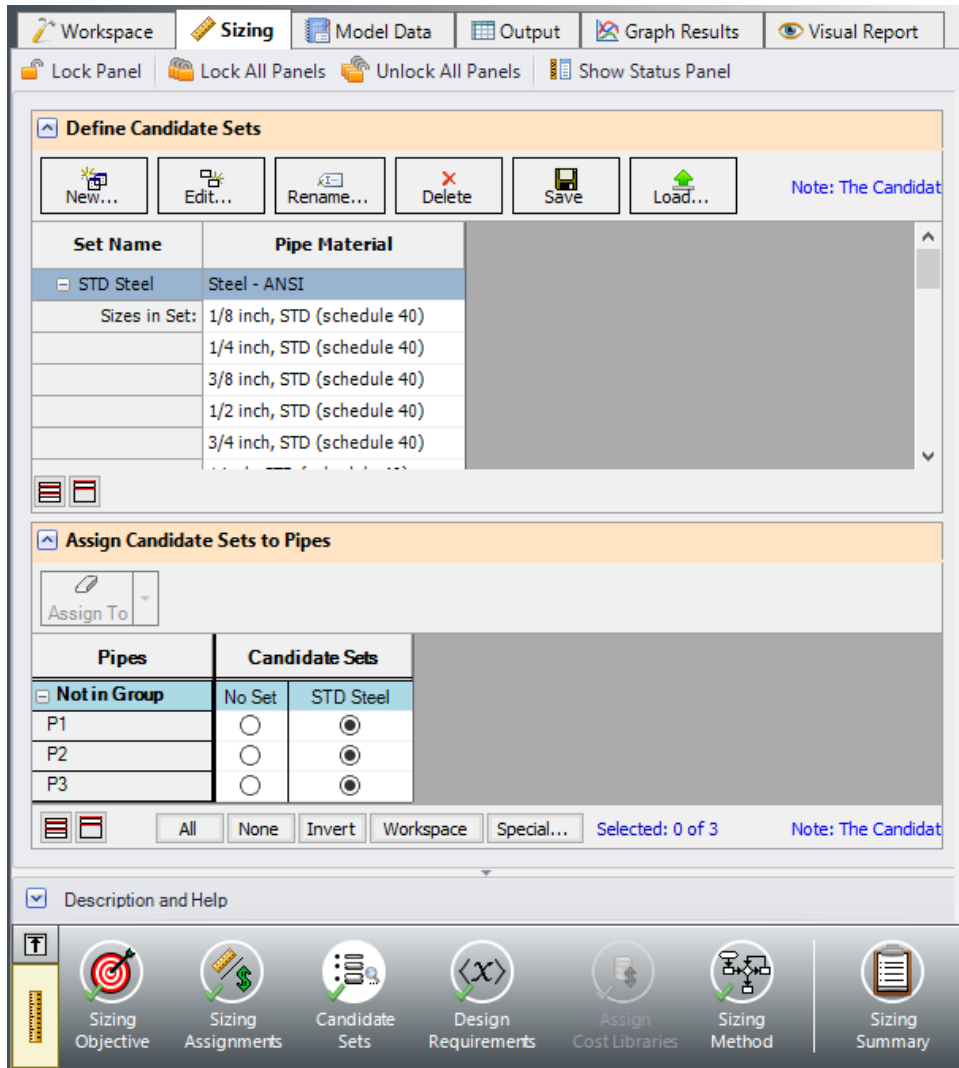


Figure 6: Candidate Sets window fully defined for the Three Tank model

### D. Design Requirements

Select the Design Requirements button. If we do not define any design requirements for this model, the ANS module will choose the smallest pipe size in the Candidate Set, since it has the smallest pipe weight. Along with the smallest pipes we may get unacceptably low flowrates, or unacceptably high velocities and pressure drops. To maintain acceptable system operating conditions, we need to set Design Requirements.

We are going to define one Design Requirement for this case. Before adding any requirements, let's look more closely at the results of the model we are starting with. Pipe 1 is 14 inch, and the other two are 12 inch. The results show that the steam flows from tank J1 to tanks J2 and J3. The mass flow rate out of J1 is 143.38 lbm/sec, and the mass flow rates to J2 and J3 are 29.07 and 114.31 lbm/sec, respectively. Let's assume that there is a requirement to deliver at least 28 lbm/sec each to J2 and J3.

We need to create a Design Requirement that represents the flow requirement. To do this:

1. Make sure that the Pipes button is selected.
2. Click New under Define Design Requirements.
3. Enter the name *Min Flow*.
4. A new row will appear in the Pipe Design Requirements table. In this row, select Mass Flow Rate as the Parameter.
5. Choose Minimum for Max/Min, and enter 28 lbm/sec

We now need to apply this Design Requirement to the pipes. In the Assign Design Requirements to Pipes section select the check box for Min Flow next to pipes P2 and P3. This will set the minimum flow rate going into Tanks J2 and J3 as described above. The fully defined and assigned Design Requirements are shown in Figure 7.

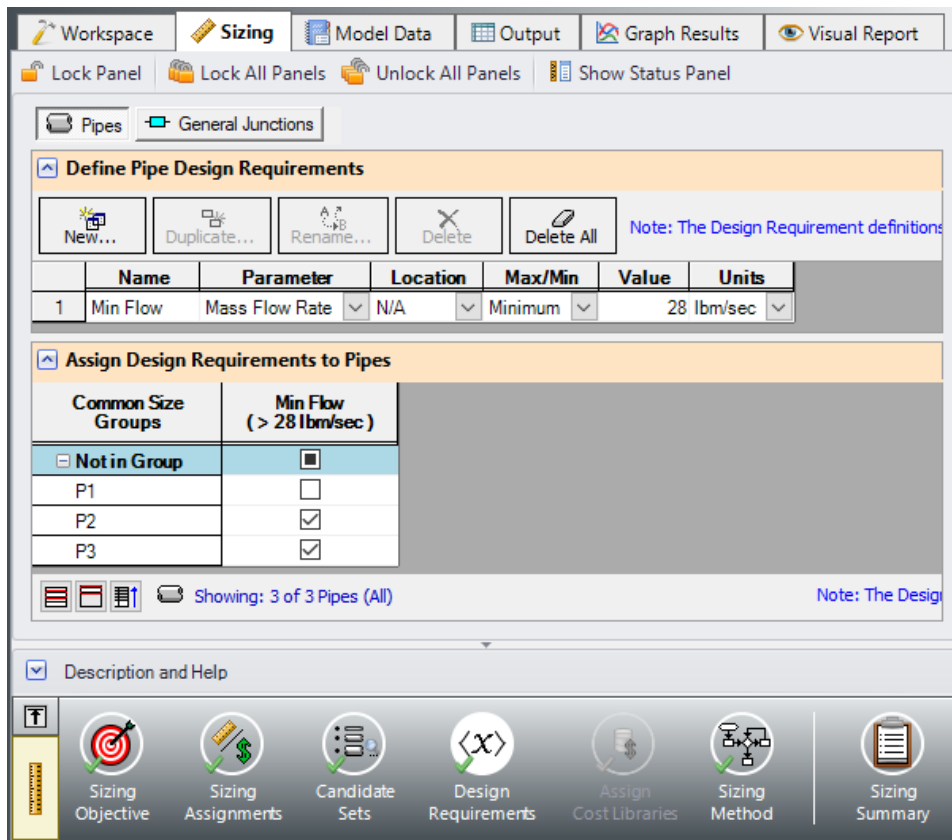


Figure 7: Design Requirements window with the minimum flow requirements defined

## 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 Library 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 Sizing Method Panel.

The Sizing Method Panel is used to set up the calculation methods for sizing the system. The User has the option to select whether discrete or continuous sizing will be used, and which method will be applied.

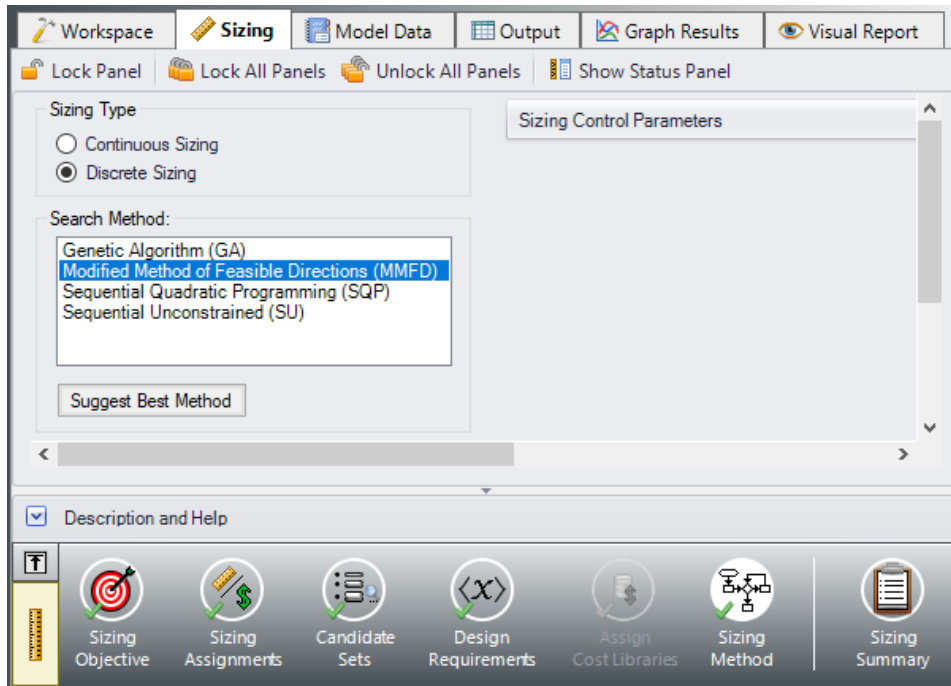
If Continuous Sizing is selected, the ANS module will use the maximum and minimum pipe sizes from the Candidate Sets as bounds, and report the ideal hydraulic diameter for the pipes between those bounds. This would result in pipe sizes which would not match any of the possible chosen commercial sizes. While this is not useful as a final solution, this can be helpful as a baseline to check the final discrete sizing solution. 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, select Discrete Sizing if it is not already selected.**

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 Best 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 two Design Requirements (the minimum flow at the two delivery pipes), and only three pipe sizes are being varied.

➤ **Choose the default Modified Method of Feasible Directions (MMFD) (Figure 8).**



**Figure 8: Sizing Method panel defined for Three Tank model**

### Step 4. Run the Model

Select Run from the Toolbar or 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 9). 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.



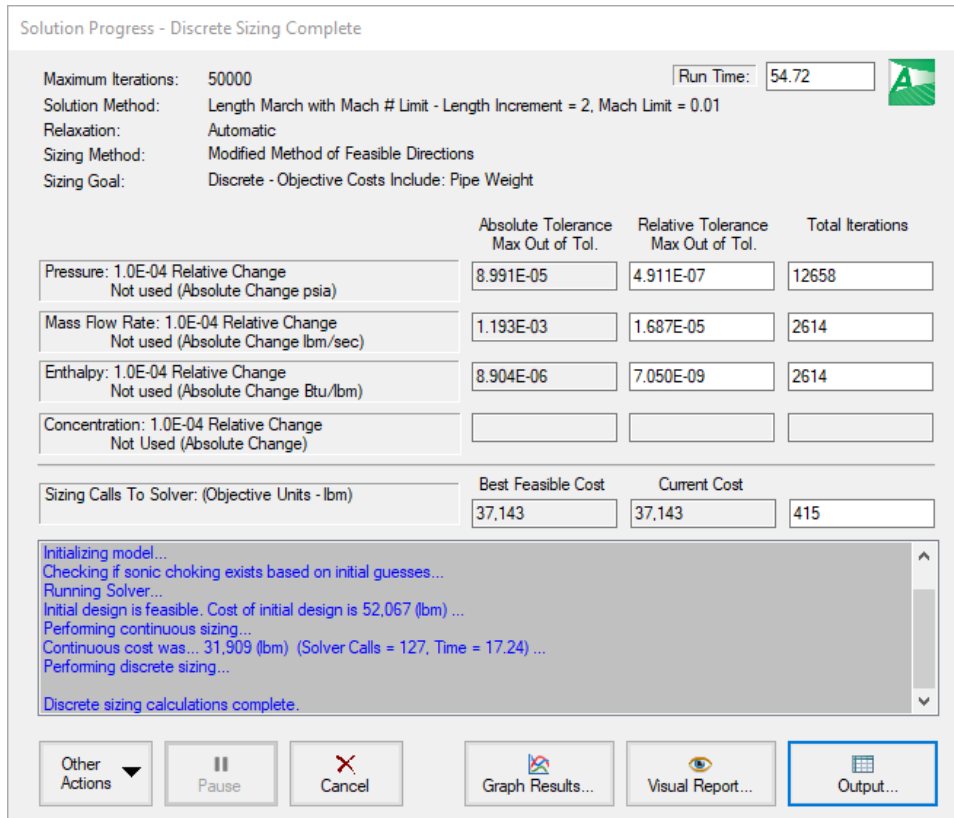


Figure 9: Solution Progress Window showing the progress of the sizing calculation

### Step 5. Examine the Output

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 10, the minimum total pipe weight identified by the ANS module was 37,143 lbm. In the Pipes section of the output on the Sizing tab, the final pipe sizes of 12, 8, and 6 inches are shown (Figure 10).

Now switch to the Pipe Design Requirements tab in the Pipe section (Figure 11). 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 2500 gpm, and the flows to J2 and J3 exceeded 750 gpm. 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 the difference between the discrete and continuous solutions was outside of a pre-defined tolerance.

You can also display the adjusted pipe sizes and resulting flow rates in the Visual Report. Options are available in the Visual Report Control which would allow the user to compare parameters such as the final pipe sizes to the original pipe sizes.

Table Units: lbm					
Type	Name	Pipe Weight			
<b>Total of All Shown Costs</b>		<b>37,143</b>			
<b>Items In Sizing</b>		<b>37,143</b>			
<b>Items Not In Sizing</b>		<b>0</b>			
<b>Pipe Subtotal</b>		<b>37,143</b>			
o	P1	Pipe	Pipe	24,782	
o	P2	Pipe	Pipe	8,566	
o	P3	Pipe	Pipe	3,795	

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (inches)
1	Pipe	Steel - ANSI	12 inch	STD	12.000
2	Pipe	Steel - ANSI	8 inch	STD (schedule 40)	7.981
3	Pipe	Steel - ANSI	6 inch	STD (schedule 40)	6.065

Figure 10: Output window with Cost Report and Pipe Sizes for Three Tank Model

Pipe	Mass Flow Rate Minimum > 28 lbm/sec [Min Flow]
2	40.59
3	30.14

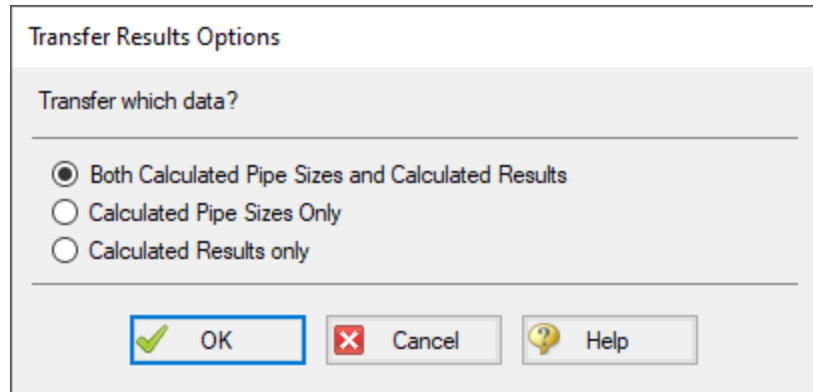
Figure 11: Design Requirements results for Three Tank Model

### 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 12. 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 12:** 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 pipe weight with the ANS module.

## Control Valve - ANS

### Summary

---

This example will show how to use AFT Arrow 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 20 psid 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 [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

In addition, it is assumed that the user has worked through the [Beginner: Three Tank Steam System - 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 Arrow installation:

- *US - Control Valve.aro*
- *US - Control Valve - ANS.aro*

---

## Step 1. Start AFT Arrow

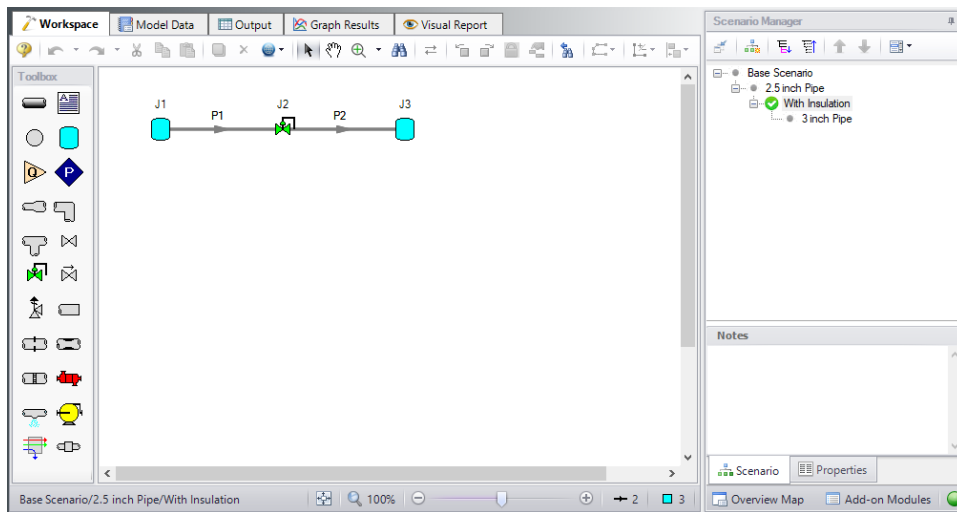
---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

Open the **US - Control Valve.aro** example file listed above, which is located in the Examples folder in the AFT Arrow application folder. Save the file to a different folder.

Make sure the "With Insulation" scenario is the active scenario by double clicking on that scenario in the Scenario Manager. The workspace should look like Figure 1



**Figure 1: Control Valve System Layout**

---

## Step 2. Define the Modules Group

---

Navigate to the Modules panel in Analysis Setup. Check the box next to Activate ANS. The Network option should automatically be selected, making ANS enabled for use.

---

## Step 3. Define the Automatic Sizing Group

---

In this model we are going to perform sizing to minimize the pipe weight (therefore minimizing cost), while maintaining a minimum pressure drop of 20 psid 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.

Set the Sizing Option as Perform Sizing.

For the Objective, choose Pipe Weight, and select the Minimize option from the drop-down list if it is not already selected.

## B. Sizing Assignments

On the toolbar 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 each of the 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. To do this:

1. Click New above Pipe Grouping, and give it new Common Size Group the name *Main Pipes*
2. A new section will now appear with the Main Pipes group listed under Common Size Groups. 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.

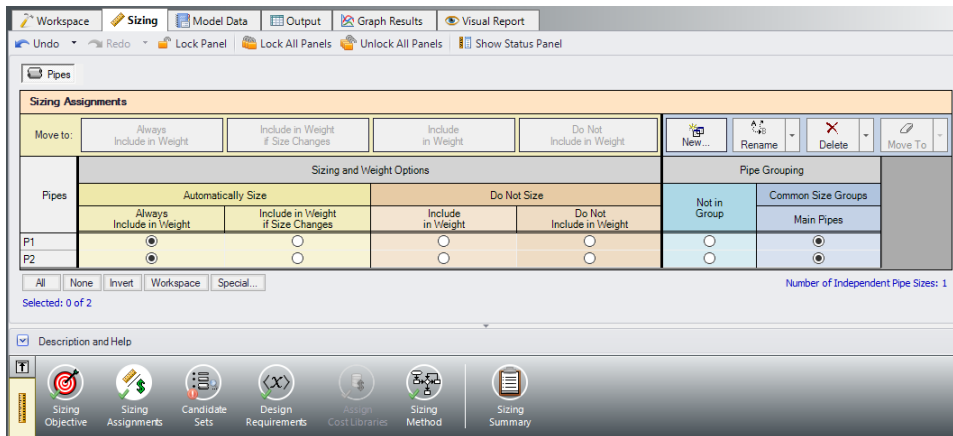


Figure 2: Pipes P1 and P2 settings in the Sizing Assignments window

## C. Candidate Sets

Click on the Candidate Sets button to open the Candidate Sets panel.

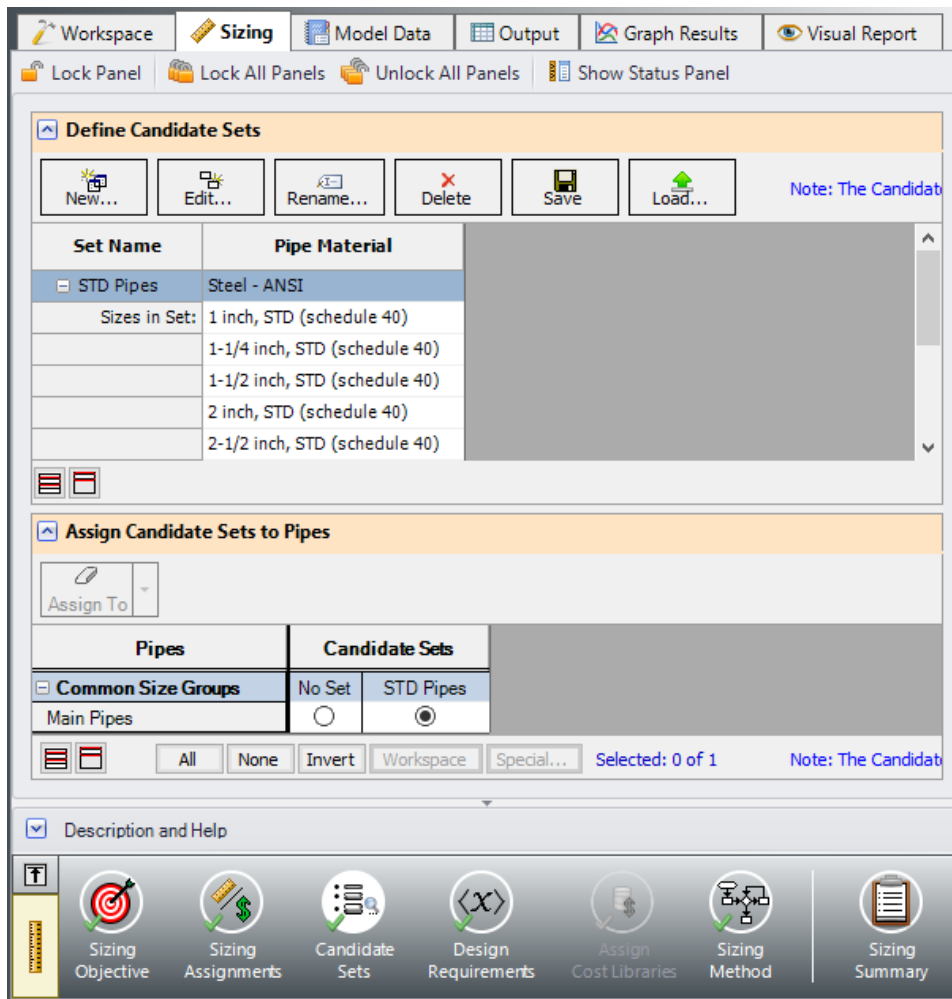
To select a discrete pipe size, ANS needs a list of candidates from which to choose. This list is called a **Candidate Set**. To create a Candidate Set, do the following:

1. Under Define Candidate Sets, click the New button.
2. Give the set the name *STD Pipes* and click OK.
3. From the drop down list choose Steel - ANSI.
4. In the Available Material Sizes and Types on the left, expand the STD pipe sizes list.
5. Double-click each of the sizes from 1 to 10 inches to add them to the list on the right.
6. In the Select Pipe Sizes window, click the OK button.

The STD Pipes set will now appear, and will be automatically defined as a Discrete set.

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 the New button under Define Control Valve Design Requirements.
3. Enter the name *Min Pressure Drop*.
4. For the Parameter, select Pressure Drop Static.
5. For the Max/Min, select Minimum
6. For the Value and Units, select 20 psid.

Now we need to apply the defined Design Requirement to the Junction.

➤ **Check the box next to the Control Valve Junction J2** in the Assign Design Requirements to Control Valves section.

---

**Note:** It is strongly recommended that flow control valves have a Design Requirement specified for the minimum pressure drop/head, or the maximum Cv/percent open at the 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 for design purposes. To avoid this behavior 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 Library panel. Since we have defined the objective as Pipe Weight, we will not need to assign any cost libraries, and this button is disabled.

## F. Sizing Method

Select the Sizing Method button to go to the corresponding panel.

Select Discrete Sizing, 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 4. Run the Model

---

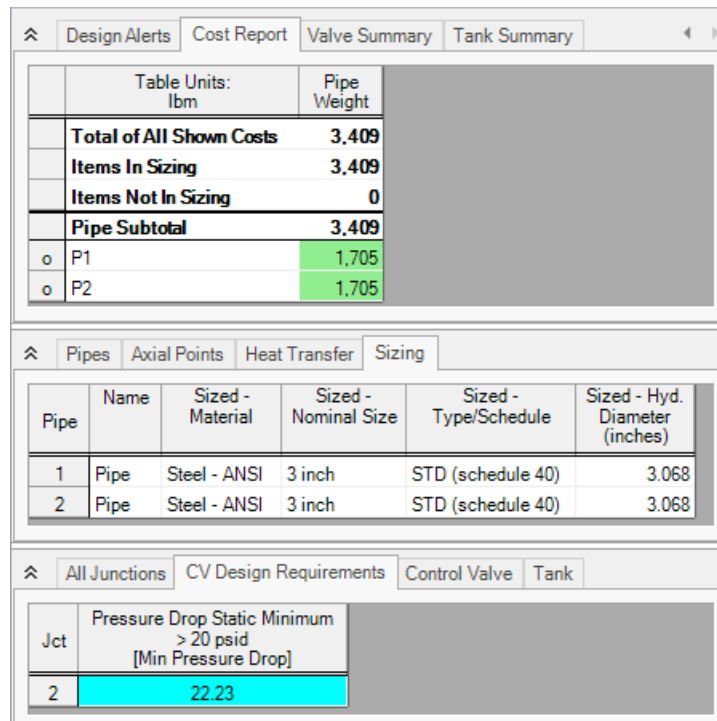
Click Run Model on the toolbar or from the Analysis menu. This will open the Solution Progress window. This window allows you to watch as the AFT Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.



## Step 5. Examine the Output

The output is shown below in Figure 4. We have shown all three sections of the output window with the General, Pipes, and Junctions sections. In the General Section, we have selected the Cost Report tab that shows the variable that was sized for this run. The pipe weight in each pipe was minimized.

In the pipe section, we have selected the Sizing tab that shows the final pipe sizes chosen. We can see that the ideal pipe size was found to be 3 inch pipe. In the Junction section, we have selected the CV Design Requirements tab that shows how we have satisfied the Design Requirement set on the control valve. The only Design Requirement was on pressure drop and we are above the minimum pressure drop of 20 psid.



**Figure 4: Output for the Control Valve model showing the Cost Report, final pipe sizes and Design Requirement for the Control Valve**

## 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.

# Air Distribution, 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 sizing for this example can only be completed if you have a license for the ANS module. However, the libraries can be built without access to ANS.

---

## 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 [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

In addition, it is assumed that the user has worked through the [Beginner: Three Tank Steam System - 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 Arrow installation:

- **US - Air Distribution - ANS Initial.aro**
- **US - Air Distribution - ANS Final.aro**
- **Air Distribution.dat** - engineering library
- **Air Distribution Costs.cst** - cost library for **Air Distribution.dat**
- **Steel - ANSI Pipe Cost.cst** - cost library for **Steel - ANSI** pipes

## Step 1. Start AFT Arrow

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

Open the **US - Air Distribution - ANS Initial.aro** example file listed above, which is located in the Examples folder in the AFT Arrow application folder. Save the file to a different folder.

## Step 2. Create an Engineering Library

---

If it is desired to size any of the junctions in the system, such as the Compressors/Fans, the first step is to create an Engineering Library containing information for each of the components that will be sized. This engineering library can later be used to attach a cost library for the components. Engineering libraries can also be connected to future models to reuse any of the components in future systems.

To add a defined compressor to an engineering library:

1. Define the Compressor/Fan junction in the workspace.
  1. For this example, the junction is already defined in the initial model.
2. Select the Compressor/Fan, go to the Library menu, and choose Add Junction to Library.
3. Give the Compressor/Fan a meaningful name such as *Fixed Flow 1250 scfm* and add it to the desired library by selecting it from the list, in this case the LOCAL USER LIBRARY.

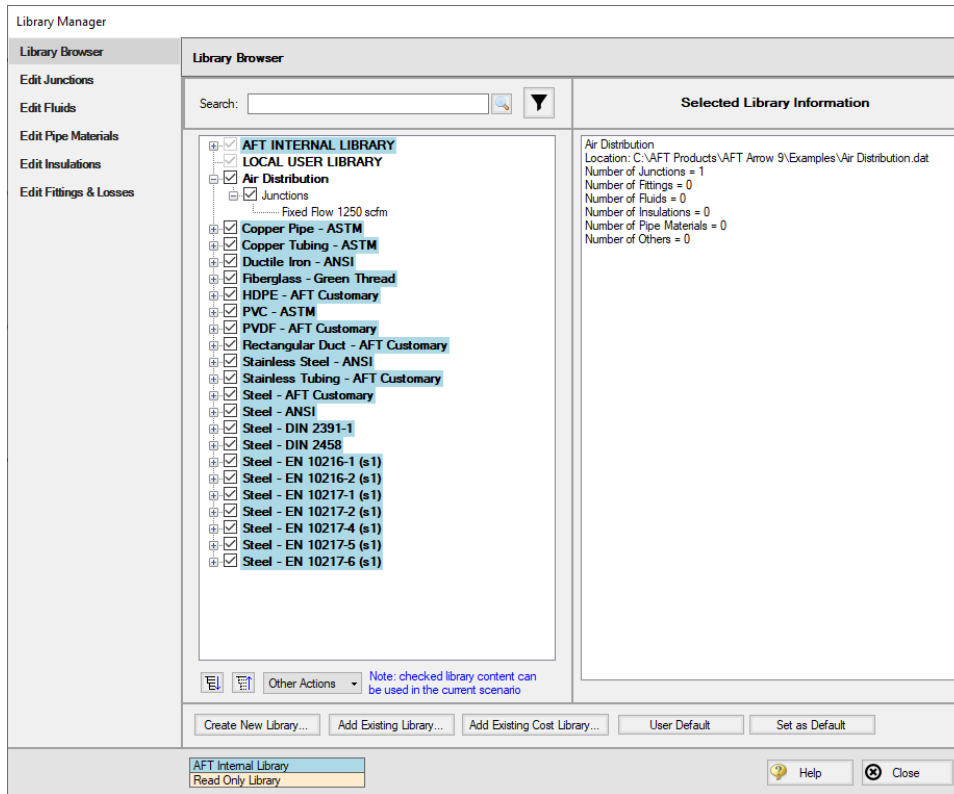
If we had multiple Compressor/Fan models, valve types, etc. then we might need to add multiple Compressors/Fans, valves, etc.

The local user library is the default location where user defaults are placed for the local machine. If the junctions are stored in the local user library, it is possible to run all 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. Therefore, it is often preferred to place the components into an external library. Follow the steps below to move the components:

1. Open the Library Manager from the Library menu.
2. Click Create New Library at the bottom of the window.
3. Give it a unique filename, such as ***Air Distribution.dat***. It is important to use a descriptive name to avoid confusion later if the library needs to be reconnected or sent to someone else.
4. Enter a Library Description. This is the name that will be displayed within the Library Manager itself and may be different from the file name. It is helpful to make the description the same name as the filename to avoid confusion. We will use the name *Air Distribution* for this library.
5. For the compressor that we added earlier, right-click the name, then click Move Content to move the junction from the LOCAL USER LIBRARY to the Air Distribution library. The Fixed Flow 1250 scfm compressor should now be visible in the Junctions section below the Air Distribution library name, as is shown in Figure 1.
6. Select Close to complete the process. If you open the folder where the library was created in step 3, the library file should now be available.

**Note:** The Copy option is also available if you want to preserve the items within the original library.

We are now ready to enter the cost data for the junctions in the model.

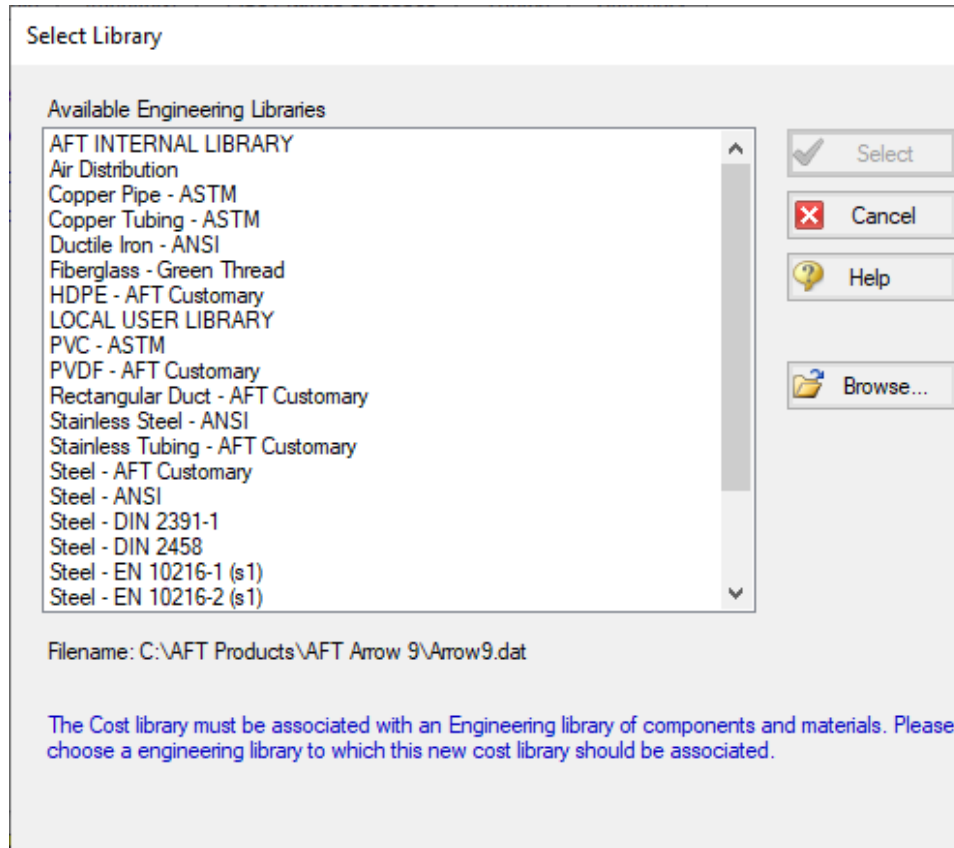


**Figure 1: Library Manager showing the components moved to the new external library from the local user library**

### Step 3. Create a Cost Library for Junctions

Use the Library menu to select Cost Library and open the Edit Cost Library window, which is used to create and view cost libraries. 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 2. Every cost library must be built of an existing engineering or pipe material library in order to link the cost information with the appropriate components in the model. Select to the newly created library, Air Distribution. You may need to use the Browse button to find it if the library is not shown in the list. Then click Select.



**Figure 2: The Available Engineering Libraries list which can be used to build a cost library**

From here we will be brought to the General tab of the Edit Cost Library window. The description field is used as the name displayed within the Library Manager, similar with the engineering library.

The Notes field can be used to provide information on sources for the data, or other details which may be useful for tracking purposes and can be seen when reviewing the library contents in the Library Manager.

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 *Air Distribution Costs***, as seen in Figure 3. Now let's move on to entering cost data. Click the Junctions tab to view the Junctions overview.

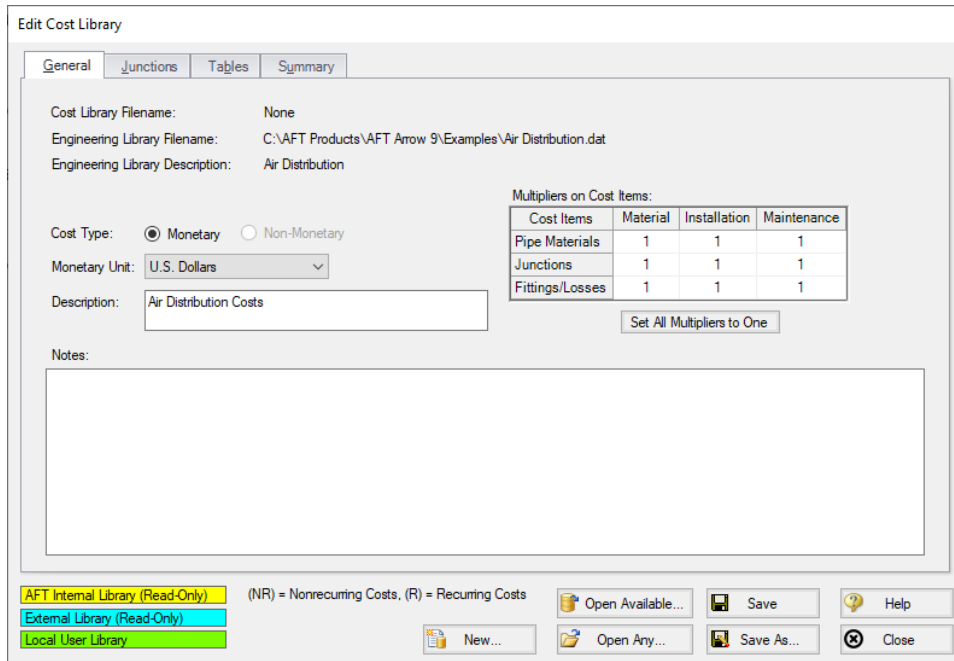


Figure 3: The General tab within the Cost Library window

Within the Junctions tab each individual library item can be viewed under each Junction type. First expand the Compressor/Fan section by clicking the + icon and select the Fixed Flow 1250 scfm component that was added earlier.

**Note:** The Show Engineering Data button can be used to view the properties window for the selected component in the engineering library.

➤ **Click the New Cost button** 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.

In the file explorer open the excel spreadsheet titled **Air Distribution Costs - ANS.xlsx** found in the Examples folder, and go to the US - Junction Costs tab. The cost data for the compressor can be seen here, categorized by cost type.

In this case we are sizing the compressor, so rather than having a cost for a specific compressor model, we have a range of costs specified based on the power requirement for the compressor. We will need to enter this data into the Tables tab first before we can define the cost item on the Junctions tab, so click on the Tables tab at the top of the window.

➤ **Click the New Table button** to begin defining the data table, and name it *Compressor 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 4.

The data we have available is the cost for the compressor over a range of different power requirements, so choose Power and Cost as shown below, then click OK.

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 compressor material, then click the Edit Table button and choose Paste. Make sure that the table Units have been set to hp. Repeat this process for the compressor installation costs using the data from the spreadsheet as shown in Figure 5.

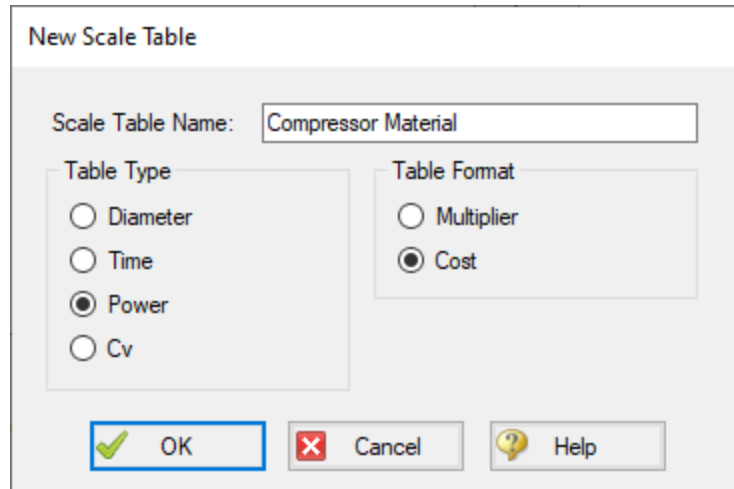


Figure 4: Table type and formatting options when creating a cost library

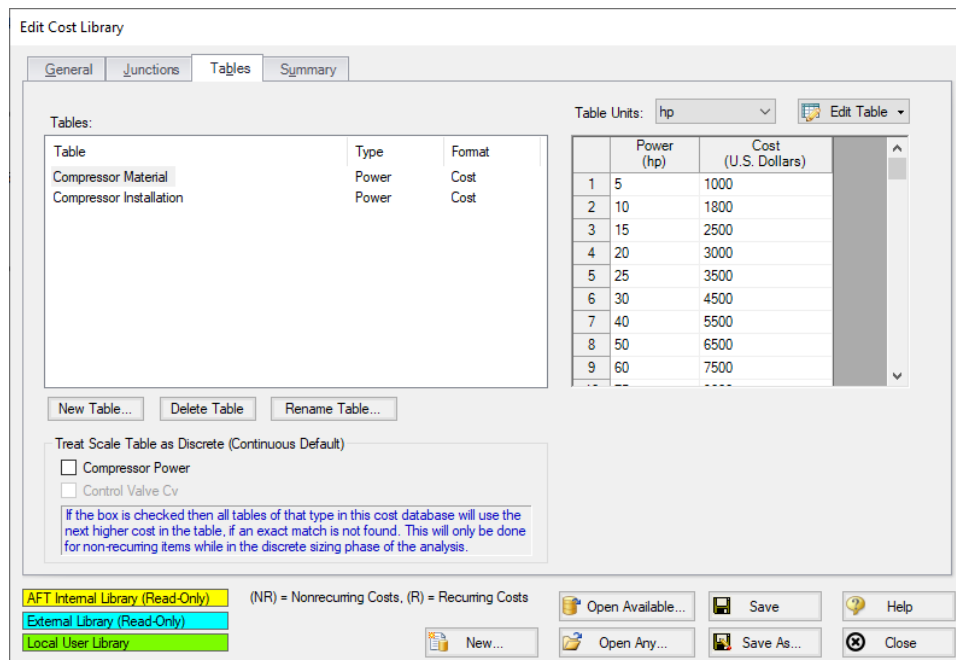
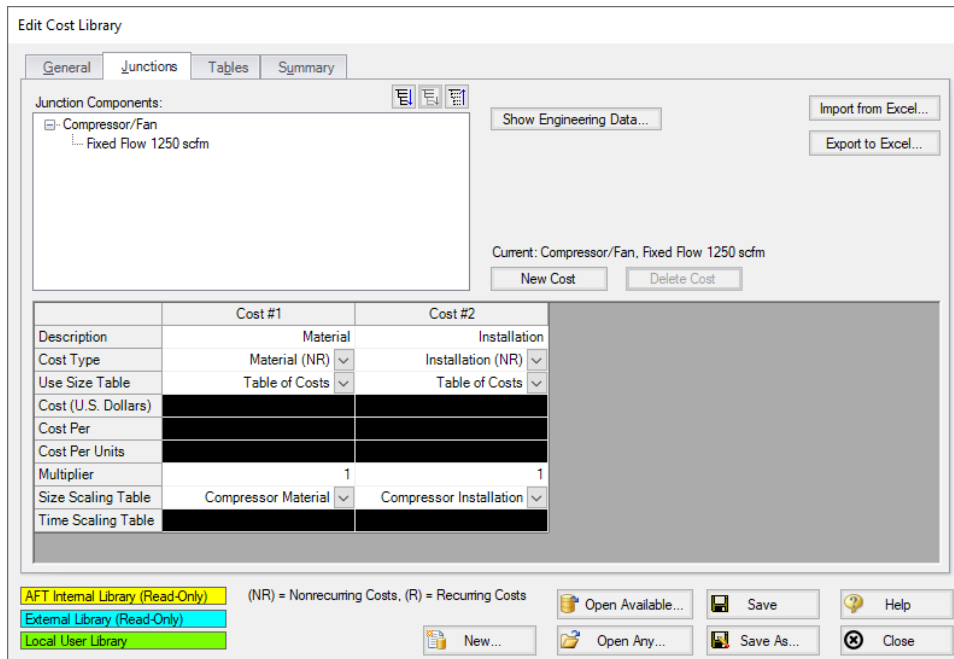


Figure 5: Complete tables list with material and installation cost tables for each of the junctions being analyzed

Now that the data tables have been created, we must assign them to the Compressor/Fan junction in the Junctions tab. Click on the Junctions tab and make sure that the Fixed Flow 1250 scfm compressor model has been selected, then enter the cost information as follows:

1. For Cost #1 in the Description row, enter the name *Material*. This row is optional, but can be useful to provide clarification on the source of the entered cost.
2. The Cost Type should be specified as Material (NR), which is the default option. Note that NR denotes a non-recurring cost type, while R denotes a recurring cost type.
3. We want to use the Compressor Material table specified earlier, so next to Use Size Table select Table of Costs.
4. Select the Compressors Material table as the Size Scaling Table. Cost #1 is now complete.
5. Click New Cost to create a second cost item for the Compressor, and name it *Installation*.
6. Choose Installation as the cost type.
7. Specify Table of Costs for Use Size Table, then set the Size Scaling Table to the Compressor Installation table. The window should now appear as shown in Figure 6.

Click Save and give the cost library a name. We will use ***Air Distribution 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.



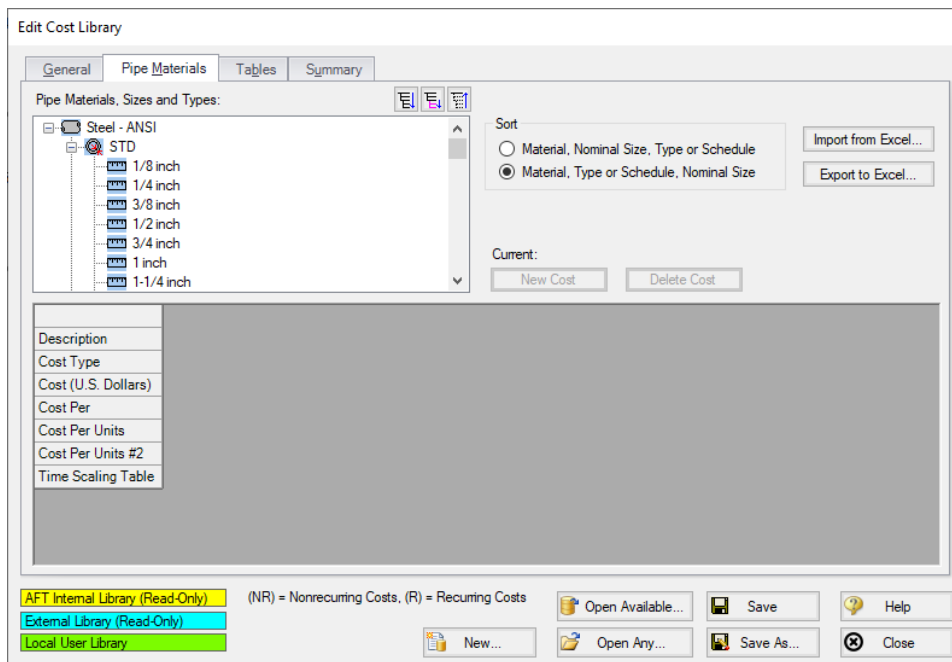
**Figure 6: Compressor Material and Installation costs assigned using cost tables**

#### Step 4. Create a Cost Library for Pipes

We now have costs for the equipment, but we still need costs for the pipes. Cost data for Steel-ANSI pipe from 1 to 20 inches are provided in the Excel spreadsheet opened earlier.



1. Open to the Edit Cost Library window again from the Library menu.
2. Click New
3. From the Available Engineering Libraries list, click Steel – ANSI.
4. Click Select to choose this library. You should now be on the General tab of the Cost Library window.
5. In Description, give the new pipe cost library a descriptive but unique name to differentiate it from the original library created for this model, such as *Steel - ANSI Pipe Costs*.
6. Go to the Pipe Materials tab. In the list section, we will be able to view all 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.
7. Expand the Steel - ANSI list, then expand the STD list (Figure 7).



**Figure 7: Pipe Materials tab in the Cost Library window showing the properties for the currently selected pipe**

If a size from the list is selected, the hydraulic diameter can be viewed, along with a summary of the material, size and type. The New Cost button will also become active (Figure 7). 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 the button labeled Export to Excel on the right side of the window. The Excel Export window should be shown as in

Figure 8, 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 Arrow 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 9.
4. Open the **Air Distribution Costs - ANS.xlsx** spreadsheet with the costs for this example. Browse to the Pipe Costs tab and copy the Material and Installation costs for the 1 inch to 20 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 20 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. When this information is added back into the model where lengths are in meters, Arrow ANS will convert the cost/length information appropriately.
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.

---

Material Name	Size	Type	Material	Installation	Maintenance
Steel - ANSI	1/8 inch	STD			
Steel - ANSI	1/4 inch	STD			
Steel - ANSI	3/8 inch	STD			
Steel - ANSI	1/2 inch	STD			
Steel - ANSI	3/4 inch	STD			
Steel - ANSI	1 inch	STD			
Steel - ANSI	1-1/4 inch	STD			
Steel - ANSI	1-1/2 inch	STD			
Steel - ANSI	2 inch	STD			
Steel - ANSI	2-1/2 inch	STD			
Steel - ANSI	3 inch	STD			
Steel - ANSI	3-1/2 inch	STD			
Steel - ANSI	4 inch	STD			
Steel - ANSI	5 inch	STD			
Steel - ANSI	6 inch	STD			
Steel - ANSI	8 inch	STD			
Steel - ANSI	10 inch	STD			
Steel - ANSI	12 inch	STD			
Steel - ANSI	14 inch	STD			
Steel - ANSI	16 inch	STD			

**Figure 8: Excel Export window showing pipe items that will be included in the excel sheet**

	A	B	C	D	E	F
1	Steel - ANSI Pipe Costs					
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 inch	STD			
7	Steel - ANSI	1-1/4 inch	STD			
8	Steel - ANSI	1-1/2 inch	STD			
9	Steel - ANSI	2 inch	STD			

**Figure 9: Initial excel sheet after exporting template for the Steel - ANSI Pipe Costs**

Back in AFT Arrow in the Edit 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 Arrow, 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 10. Click Save to save the cost library with a unique file name (i.e. **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.

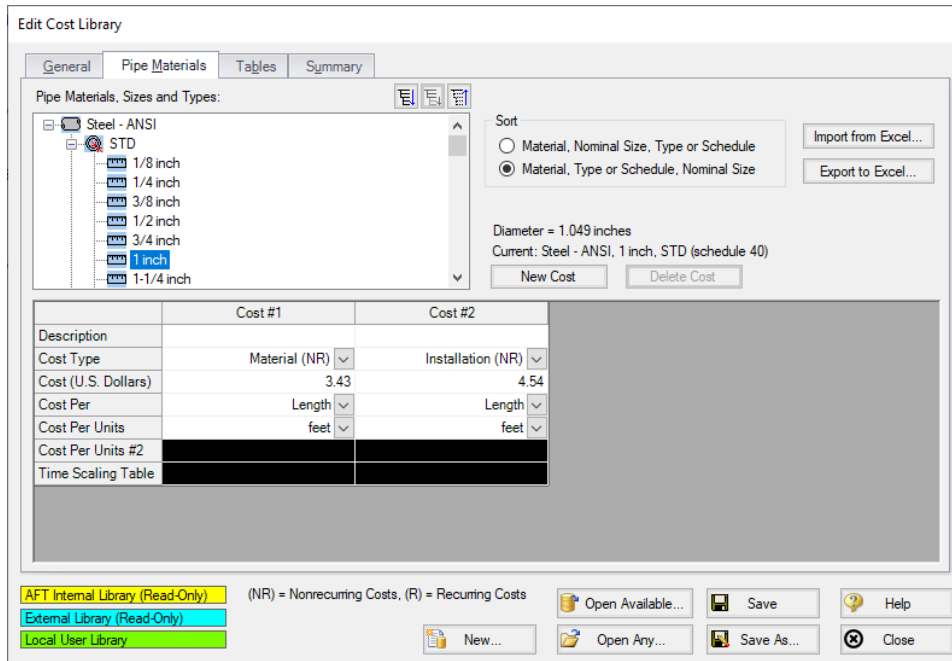


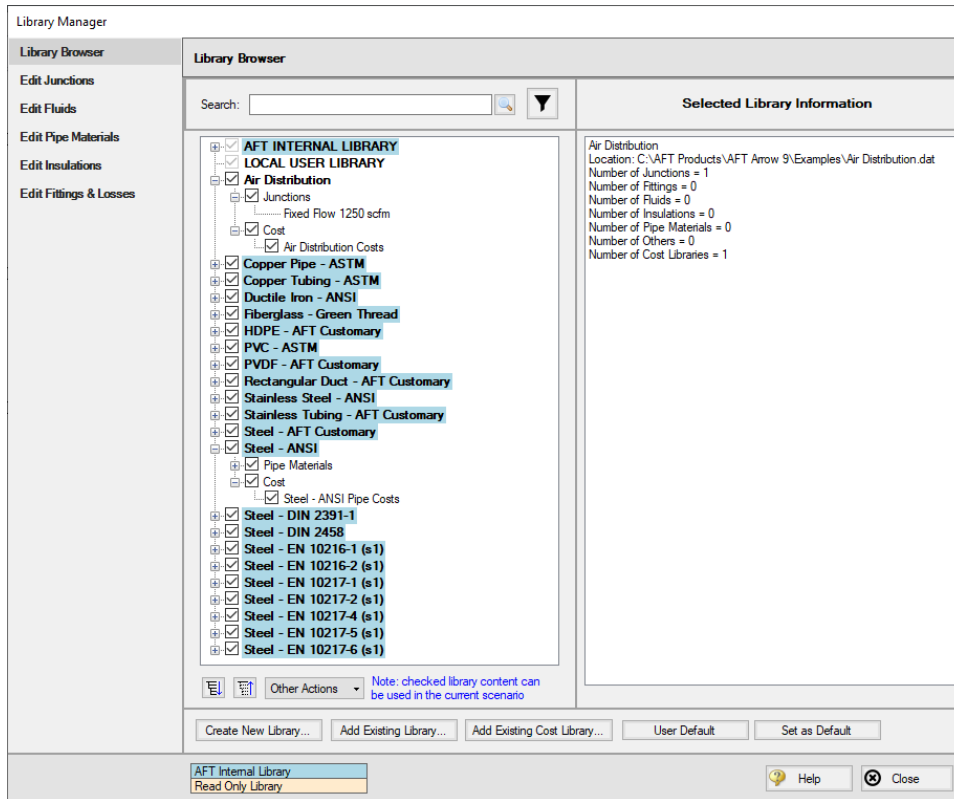
Figure 10: The Pipe Materials tab in the Cost Library window, showing the data recently imported

## Step 5. Connect the Libraries

Open the Library Manager from the Library menu. Currently, the engineering library and two cost libraries should have automatically connected to the current model. In case they are not, we will now need to connect the libraries so that the junctions in the model will be linked to the costs and junctions in the engineering and cost libraries.

If the Air Distribution engineering library, Air Distribution Costs library, and Steel - ANSI Pipe Costs library are not connected to the model, follow the steps below to connect them.

➤ **Choose Add Existing Library** at the bottom of the Library Manager window, then browse to **Air Distribution.dat**. Next choose **Add Existing Cost Library** and browse to **Air Distribution Costs.cst** then **Steel - ANSI Pipe Costs.cst**. This should make the libraries available, and automatically connect them, as shown in Figure 11.



**Figure 11: The Library Manager with the new cooling system engineering library and its associated equipment costs**

## Step 6. Define the Automatic Sizing Group

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

Go to the Sizing Objective panel. For the Objective, select Monetary Cost then select Calculate Cost, Do Not Size.

In the Options to Minimize Monetary Cost section, select the radio button to include Energy in Cost Report Only. The System Life and Energy Cost sections should now become available. Enter a System Life of 5 years and an Energy Cost of 0.11 U.S. Dollars Per kW-hr.

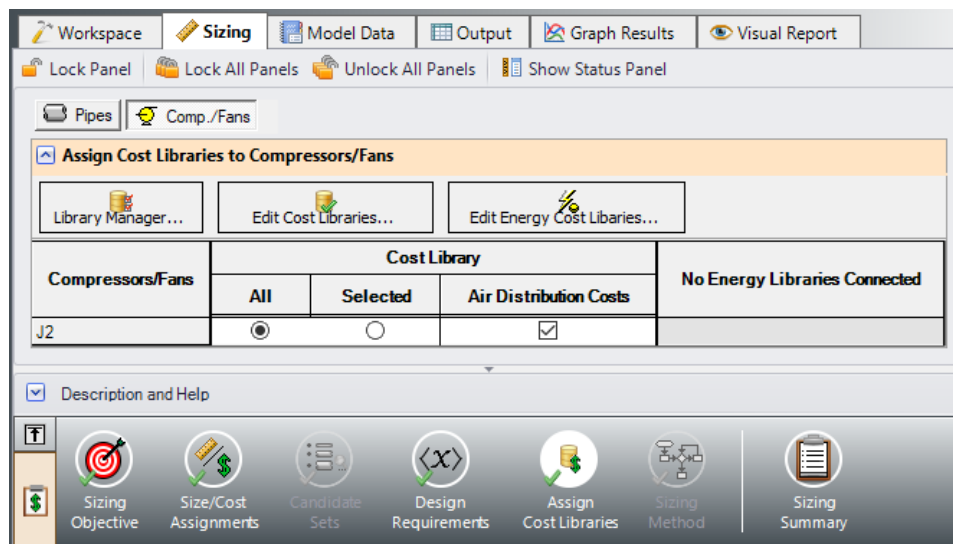
### B. Size/Cost Assignments

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 Comp./Fans button at the top to go to the Compressors/Fans size/cost assignments page. Select the radio button for J2 to Include in Cost Report Only.

### C. Assign Cost Libraries

Go to the Assign Cost Libraries panel. All 18 pipes should be assigned to the Steel - ANSI Pipe Costs library. Go to the Comp/Fans page next and select Air Distribution Costs, as shown in Figure 12



**Figure 12: Applying the newly built cost library for the compressor on the Assign Cost Libraries panel**

**Note:** Before running the model, be aware that there are no design requirements set for the control valves. When the solver begins to run, a Model Diagnostics window will appear to inform you that this may result in control valves adding pressure in order to achieve the setpoint. As you will see, the control valves will not add pressure, so this message is safe to ignore.

### 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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

### Step 8. Examine the Output

On the Cost Report, the total cost should be \$178,626 as shown in Figure 13. In the Library Sources tab in the Junction section, the Compressor/Fan should list the Air Distribution Costs library as the cost

source as shown in Figure 14. The Library Sources in the Pipes section should similarly show the new pipe cost library for Material and Installation.

<span>General</span> <span>Warnings</span> <span>Design Alerts</span> <span>Cost Report</span> <span>Compressor/Fan Summary</span> <span>Valve Summary</span> <span>Tank Sur</span>						
Table Units: U.S. Dollars	Material	Installation	Non-Recurring Sub Total	Operation/Energy	TOTAL	
<b>TOTAL OF ALL MODEL COSTS</b>					<b>178,626</b>	
<b>Total of All Shown Costs</b>	<b>67,456</b>	<b>43,088</b>	<b>110,544</b>	<b>68,082</b>	<b>178,626</b>	
<b>Items In Sizing</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	
<b>Items Not In Sizing</b>	<b>67,456</b>	<b>43,088</b>	<b>110,544</b>	<b>68,082</b>	<b>178,626</b>	
<b>Pipe Subtotal</b>	<b>64,560</b>	<b>41,640</b>	<b>106,200</b>	<b>0</b>	<b>106,200</b>	
o P1	1,340	860	2,200	0	2,200	
o P2	3,350	2,150	5,500	0	5,500	
o P3	16,750	10,750	27,500	0	27,500	
o P4	8,375	5,375	13,750	0	13,750	
o P5	8,375	5,375	13,750	0	13,750	
o P6	8,375	5,375	13,750	0	13,750	
o P7	8,375	5,375	13,750	0	13,750	
o P8	8,375	5,375	13,750	0	13,750	
o P9	125	101	225	0	225	
o P10	125	101	225	0	225	
o P11	125	101	225	0	225	
o P12	125	101	225	0	225	
o P13	125	101	225	0	225	
o P14	125	101	225	0	225	
o P15	125	101	225	0	225	
o P16	125	101	225	0	225	
o P17	125	101	225	0	225	
o P18	125	101	225	0	225	
<b>Compressor/Fan Subtotal</b>	<b>2,896</b>	<b>1,448</b>	<b>4,344</b>	<b>68,082</b>	<b>72,426</b>	
J2	2,896	1,448	4,344	68,082	72,426	

Figure 13: Output tab showing the Cost Report

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	

Jct	Name	Type	Material	Installation	Maintenance	Operation/Energy
1	Atmospheric Intake	Tank				
2	Compressor/Fan	Compressor/Fan	Air Distribution Costs	Air Distribution Costs		Fixed Energy Cost
3	Branch	Branch				
4	Branch	Branch				
5	Branch	Branch				
6	Branch	Branch				
7	Branch	Branch				

Figure 14: Output tab showing the current Library Sources used in the model calculations



## Air Distribution - 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 5-year operating period
2. Size system for **life cycle** cost with a 5-year operating period
3. Size system for **life cycle cost with an initial cost limit** for a 5-year operating period

---

**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 and using Initial Cost Limits
- Sizing a compressor and sizing piping for the chosen compressor
- Interpreting Cost Reports

### Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

In addition, it is assumed that the user has worked through the [Beginner: Three Tank Steam System - 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 Arrow installation:

- **US - Air Distribution - ANS Initial.aro**
- **US - Air Distribution - ANS Final.aro**
- **Air Distribution.dat** - engineering library
- **Air Distribution Costs.cst** - cost library for **Air Distribution.dat**
- **Steel - ANSI Pipe Cost.cst** - cost library for **Steel - ANSI** pipes

## Step 1. Start AFT Arrow

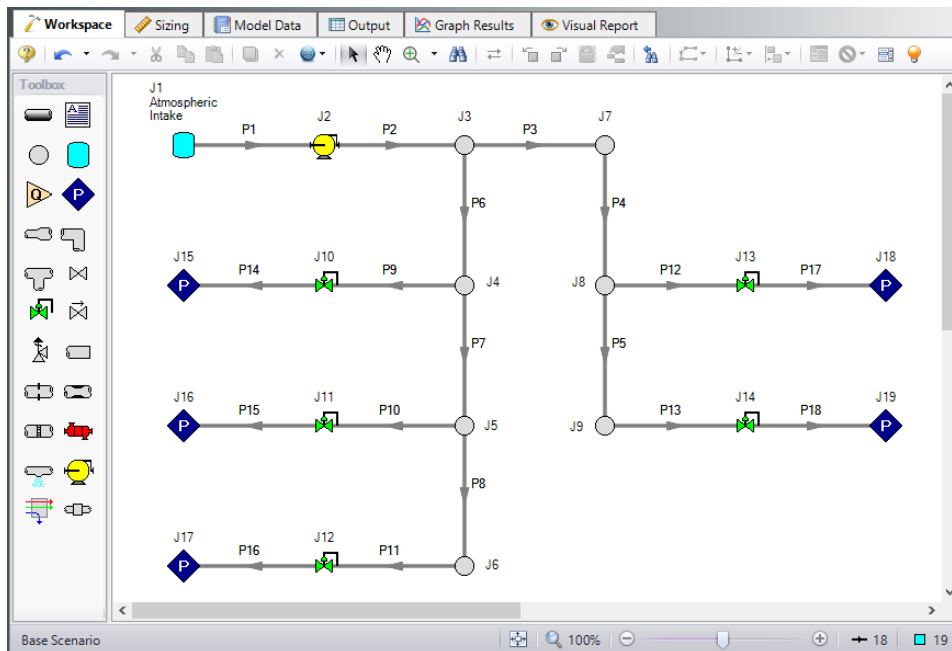
From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

Open the **US - Air Distribution - ANS Initial.aro** example file listed above, which is located in the Examples folder in the AFT Arrow application folder. Save the file to a different folder.

## Step 2. Define the Pipes and Junctions Group

The Workspace should appear as shown in the figure below.



**Figure 1: Workspace for Air Distribution model**

## Overview of Sizing Systems with Compressors

A centrifugal compressor can be modeled with either the Compressor Curve or as a fixed flow rate or pressure/head as the Sizing option. Generally, if a specific compressor has been identified, the Compressor Curve type would be chosen, whereas the Sizing option is useful during the selection stage in order to identify the compressor requirements for the system. In some cases, even though the specific compressor has not been chosen, several candidate compressors are available. For this third case it would be best to model each candidate compressor with a compressor curve using separate scenarios for each compressor.

## Sizing a Compressor

The compressor in our air distribution case is set as Sizing with a fixed flows of 1250 scfm. In order to allow the compressor to be defined as fixed flow the flow control valve at J12 has been defined at a fixed pressure drop. This model thus represents a first cut model where, based on the first cut results, the compressor will become an actual compressor with a compressor curve, and control valve J12 will become a flow control valve controlling to 250 scfm, similar to the other control valves.

As already stated, when a compressor is modeled as a fixed flow, it is not a specific compressor from a specific manufacturer. Thus, the costs for the compressor 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 ten horsepower compressor (of specific configuration and materials of construction for a given flowrate) may cost \$2700, and a forty horsepower may cost \$8250. Other typical costs for different power requirements can be approximated. The actual cost for the compressor will, of course, highly depend on the application. These costs are entered into a cost library and are accessed by the Solver. As AFT Arrow evaluates different combinations of pipe sizes, each combination will require a certain power from the compressor. With a cost assigned to this power, the ANS module can obtain a cost to enter into the objective which it sizes. Later in this chapter, we will look at the costs for the compressor in our example air distribution network.

In addition to non-recurring costs, recurring costs can also be estimated. Specifically, the cost of the power used by the compressor over a period of time (which the ANS module calls the System Life) can be determined. All AFT Arrow needs is an overall compressor efficiency to determine the actual power from the ideal power. Again, since we do not have a specific compressor selected yet, the efficiency can only be approximated. The ANS module calls this the Nominal Efficiency.

Therefore, it is recognized that at this first cut phase the values for nominal efficiency are imprecise and users may need to go through a few iterations to arrive at satisfactory results.

A step-by-step method of compressor selection proceeds in two phases. The steps are outlined in the Help System.

For this example, we will be completing the compressor selection process considering the objective of minimizing initial cost, then minimizing life cycle cost.

### Step 3. Define the Automatic Sizing Group

---

The model is already defined for a regular AFT Arrow run, but we need to complete the sizing settings before running the analysis. Go to the Sizing window by clicking the Sizing tab.

#### 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. Select Perform Sizing for this calculation.
2. For the Objective, choose Monetary Cost and select the Minimize option from the drop-down list if these options are not already selected.
3. Under Options to Minimize Monetary Cost, choose 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.
4. Set the System Life to 5 years with 0% interest and inflation rates.
5. For the Energy Cost area choose Use this Energy Cost Information, with a cost of 0.11 U.S. Dollars Per kW-hr. Note this assumes the pump will run all day, every day, for the entire 5 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.

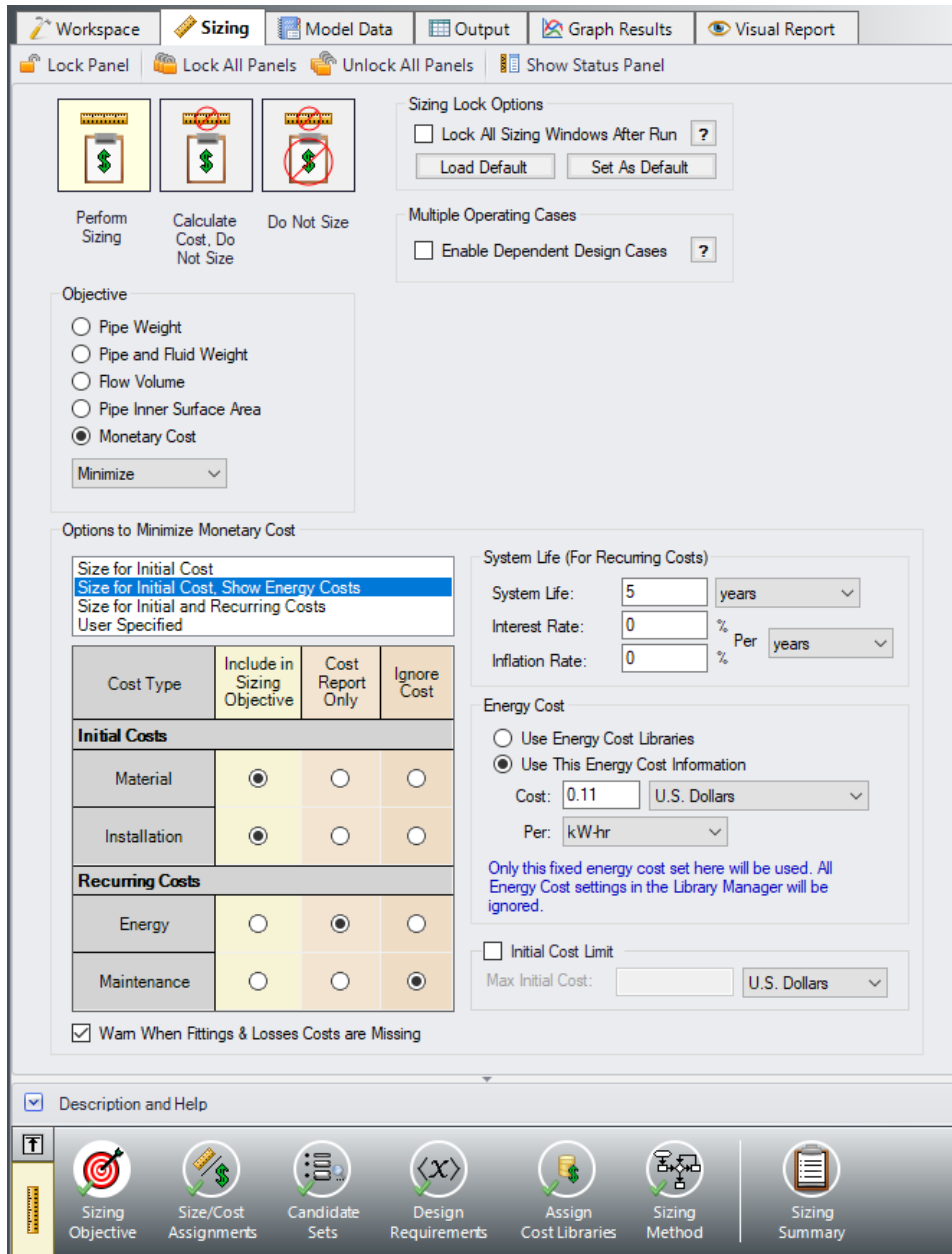


Figure 2: Sizing Objective panel setup for the initial cost analysis

## B. Size/Cost Assignments

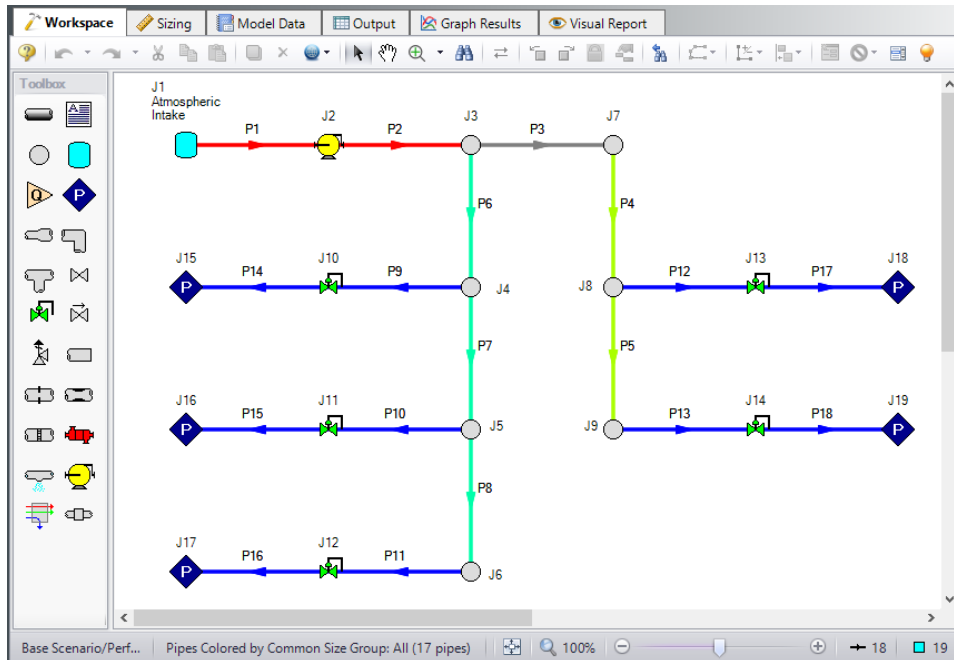
On the Sizing Navigation panel select the Size/Cost Assignments button. For this example we are sizing both the pipes and compressors for a new system, so we will need to calculate and minimize cost for all related components in the model.

➤ **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 18 pipes are selected to be Not In Group, then 18 separate pipe sizes will be selected by the automatic sizing engine. That could potentially mean, for example, different pipe diameters in pipes P9 and P14. If one wanted pipes P9 and P14 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 discharge room piping will be put into a Common Size Group. What this means is that there will be a single diameter chosen for this group of pipes, rather than potentially six different diameters.

For this system we will define four 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 these four groups.



**Figure 3: Workspace with Common Size Groups shown in different colors**

First create a Compressor size group for pipes P1 and P2, which are highlighted in red.

➤ **Click New above the Pipe Grouping section** to create the group, and name it Compressor. Select the radio buttons under the Compressor heading next to P1 and P2 to add them to the group. Alternatively, the pipes can be selected in the Workspace, then moved to a group by right-clicking the Workspace and selecting the corresponding option from the menu.

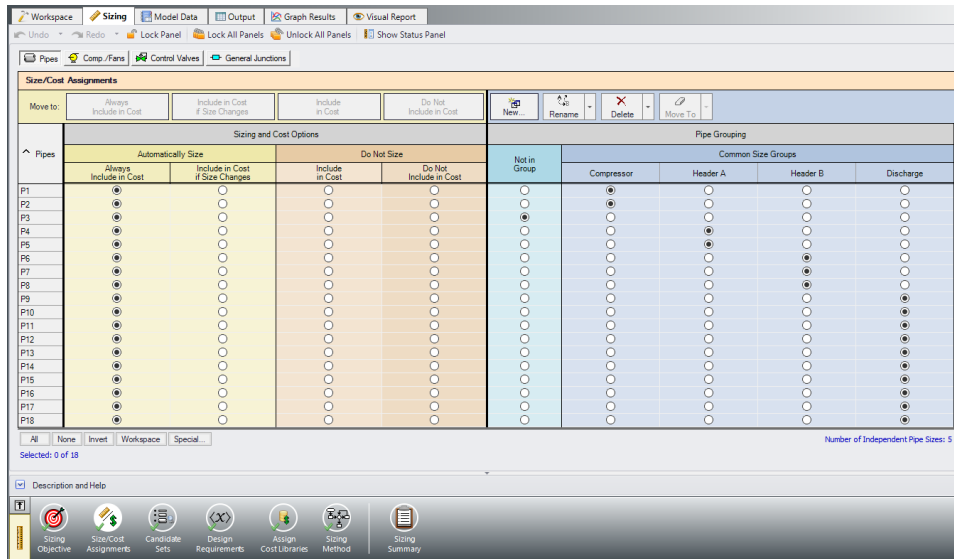
Repeat this process for the header piping. We will assume the header will have the same size, so therefore these pipes will have a common size. Like before, create a new group called Header A, and add the two header pipes on the right side of the system to the new group (pipes P4 and P5).

The headers supplying each room are one physical pipe in the field, represented as multiple pipes in AFT Arrow by necessity. Create two more groups and add the pipes as listed below and shown in Figure 4.

1. Header B = P6 - P8
2. Discharge = P9 - P18

The completed pipe Size/Cost Assignments can be seen in Figure 4. Note that pipe P3 should be the only pipe not in a group.

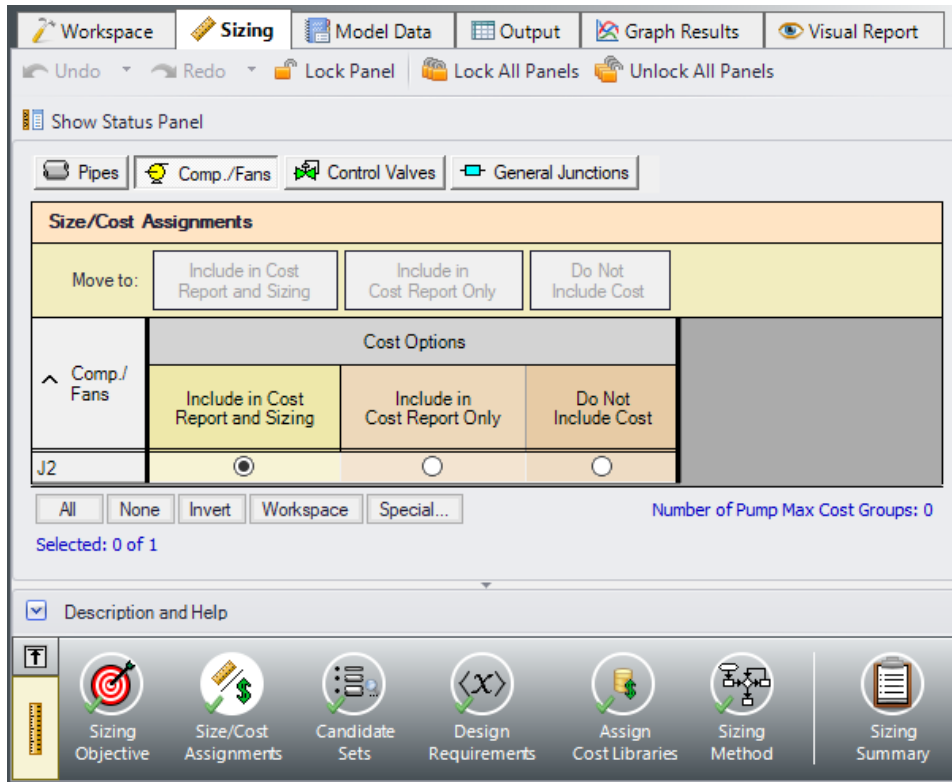
**Note:** Pipes in the Size/Cost Assignments table can be sorted by Common Size Group instead of sorting by number by clicking on the Pipe Grouping header. Clicking on the header for a particular group will sort the pipes in that specific group to the top of the window.



**Figure 4: Size/Cost Assignments panel with Common Size Groups defined as outlined above**

➤ **Click the Comp./Fans button** to switch to the Compressor/Fan Size/Cost Assignments. Select the option to Include in Cost Report and Sizing for the compressor J2 as shown in Figure 5. The compressor will now be sized concurrently with the pipes.

Similar to the Common Size Groups for pipes, compressors can be placed into Maximum Cost Groups based on monetary cost. If compressor costs are added to a Maximum Cost Group, the ANS module will determine the compressor in the group with the maximum costs, and set all compressors in the group to use the largest cost. In this case there is only one compressor, so this option is not available.



**Figure 5: Size/Cost Assignments panel defined for the compressor**

Now go to the Workspace window. It is possible to inspect the Common Size Groups in the model by coloring the pipes. From the View menu, select All under Color Size Groups. The model should appear as shown in Figure 3, though the colors may be different.

### C. Candidate Sets

Click on the Candidate Sets button to open the Candidate Sets panel.

Create a new Candidate Set for all Steel - ANSI pipes between 2" and 10" in diameter. Click the New button, and name the Candidate Set *STD Steel 2" - 10"*. Choose Steel - ANSI as the pipe material library, STD as the Material Type, and add all pipes from 2" to 10" in size.

In the bottom section of the window, check that each of the Common Size Groups and pipe P3 have been assigned to use the existing Candidate Set. The window should appear as shown in Figure 6.



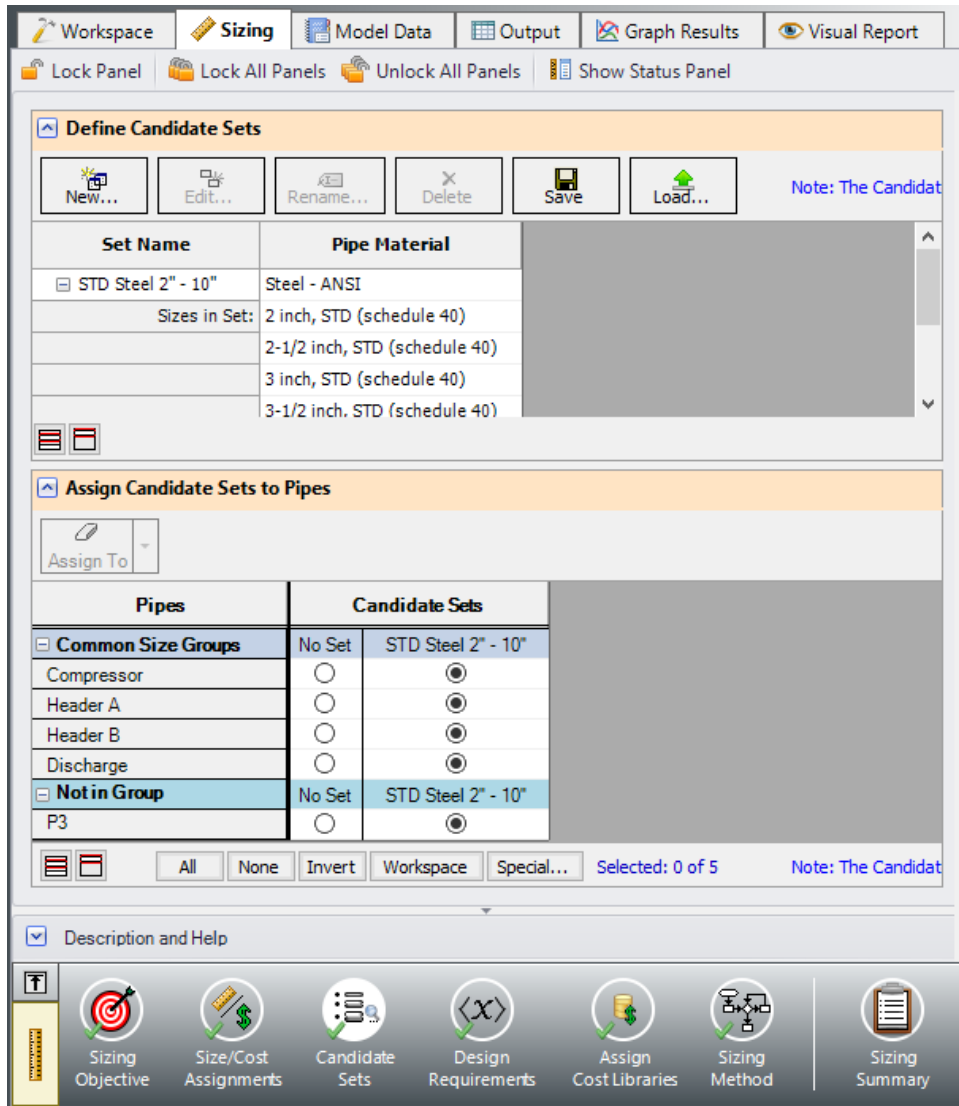


Figure 6: Candidate Sets panel fully defined for the model

### D. Design Requirements

Select the Design Requirements button from the Navigation Panel.

For this model we have multiple requirements for the system:

- All pipes have a maximum velocity of 150 feet/sec
- All control valves must have a minimum pressure drop of 2 psid
- All discharges must receive a minimum flow of 250 scfm

Define and assign the first requirement listed above as a Pipe Design Requirement as shown in Figure 7, and apply it to each of the pipes.

Notice that we have accounted for the minimum flow requirement in this system by using flow control valves in each discharge line. By using assigned flow compressor and flow control valves we have incorporated the necessary flow rate as a boundary condition, so we do not need to apply a Design Requirement to account for the second item.

For flow control valves J10, J11, J13, and J14, define and apply the minimum pressure drop requirement in the Control Valves section as shown in Figure 8. Note that for compressor sizing purposes as discussed earlier control valve J12 has been defined as a Constant Pressure Drop valve, so the pressure drop Design Requirement does not need to be applied to this valve.

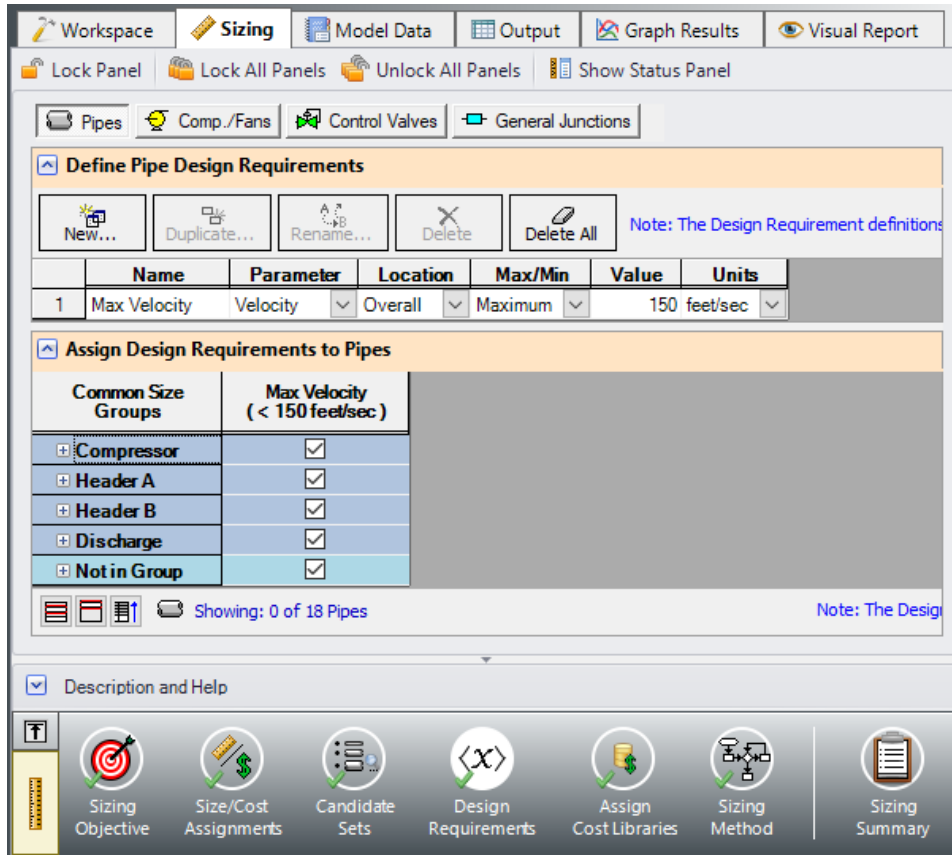


Figure 7: Design Requirements panel for pipes

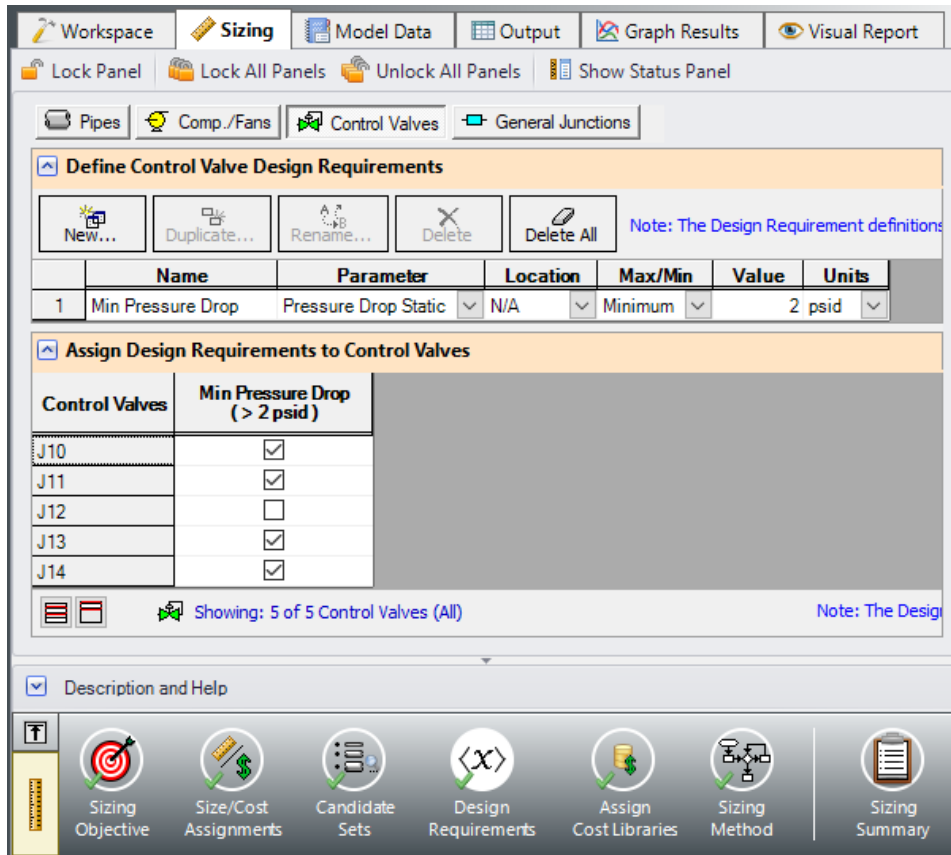


Figure 8: Design Requirements panel defined for the control valves

## 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 [Creating Libraries for Sizing a System](#) 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 compressor curve data for Step 10 of this example.

➤ **Open the Library Manager** by clicking the Library Manager button to see which libraries are connected. The AFT INTERNAL LIBRARY, LOCAL USER LIBRARY, and Pipe Material libraries should be available and connected by default. Available libraries will be shown based on libraries that Arrow 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, ***Air Distribution.dat***, which contains the data for the compressor. There are also two cost libraries, ***Air Distribution Costs.cst*** and ***Steel - ANSI Pipe Costs.cst***, which contain the cost data for the junctions and the pipe materials, respectively. ***Air***

***Distribution Costs.cst*** was built off of the engineering library ***Air Distribution.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 the button to Add Existing Library.
2. Browse to the AFT Arrow 9 Examples folder (located by default in C:\AFT Products\AFT Arrow 9\Examples\), and open the file titled ***Air Distribution.dat***.
3. Repeat the above steps, but choose Add Existing Cost Library to browse for ***Air Distribution 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 connect them by checking the box next to the library name for each of the three libraries.

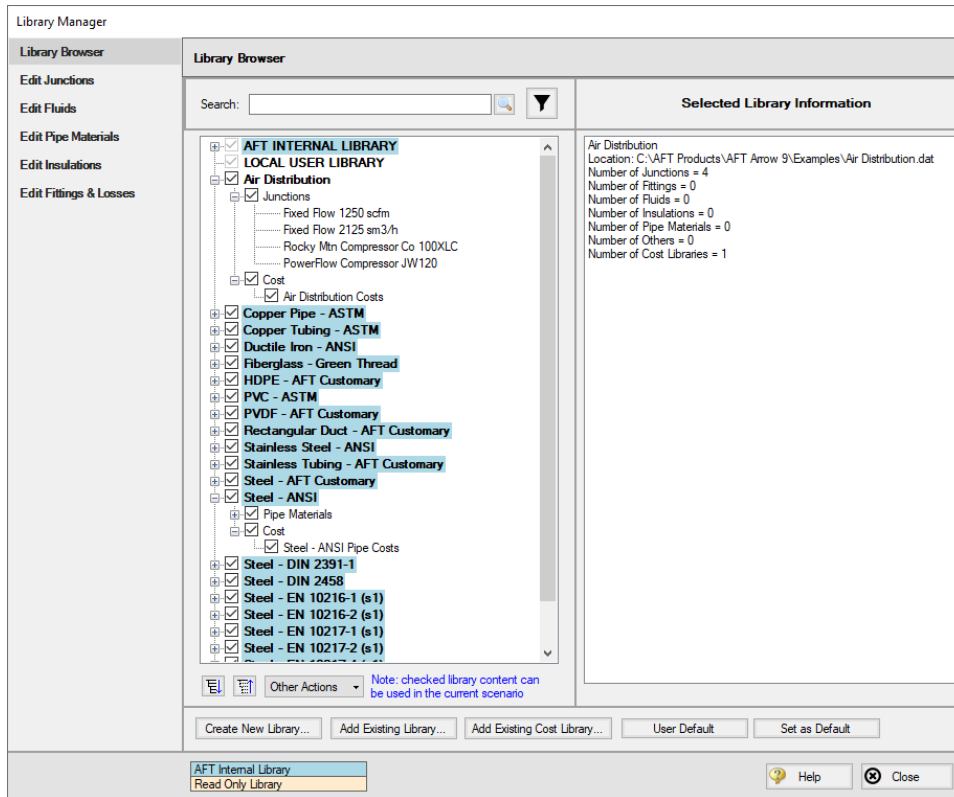
Once these steps have been completed, you should now see that the new libraries have been added to the Available Libraries list, and have been automatically connected to the model, as shown in Figure 9 for the junction libraries. The junction libraries appear with the names Air Distribution, Air Distribution Costs, and Steel - ANSI Pipe Costs.

---

**Note:** When a library is selected in the Library Manager, the filename and location can be viewed in the Selected Library Information. Library names can be different from the given filename.

---

Once you have confirmed that the necessary libraries are connected, click Close to exit the Library Manager.



**Figure 9: Library Manager with Air Distribution libraries connected**

Since we are connecting a pre-built library, we will need to take an additional step to make sure that the compressor in the Workspace is linked to the Air Distribution engineering library that we added.

1. Go to the Workspace, and open the compressor (junction J2)
2. From the Library Jct list, select Fixed Flow 1250 scfm. None of the input should change, as this library item is identical to the existing pump. However, AFT Arrow will now recognize that this junction is connected to the engineering and cost library for this compressor.

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 cost information can be assigned to it. Let's now return to the Sizing window.

Back in the Assign Cost Libraries panel for the Pipes, the Steel - ANSI Pipe Costs library should be the only visible cost library, since it is the only connected library for the pipe sizes in the Candidate Set. Make sure that the library is checked to be applied for the model as shown in Figure 10.

➤ **Click on the Comp./Fans button** to view the cost library assignments for the compressor. The only visible library should be the Air Distribution 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 the compressor (Figure 11).

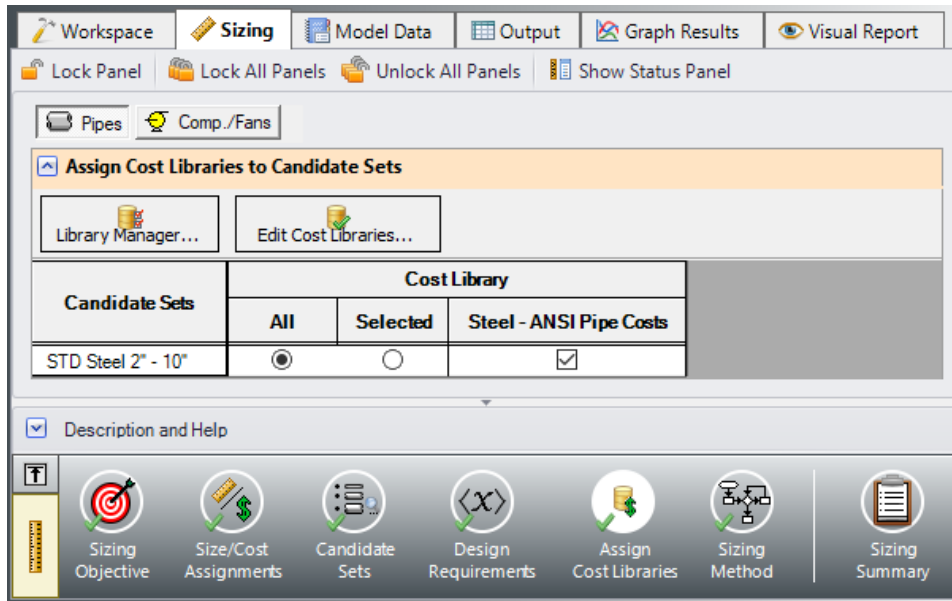


Figure 10: Assign Cost Libraries panel for pipes

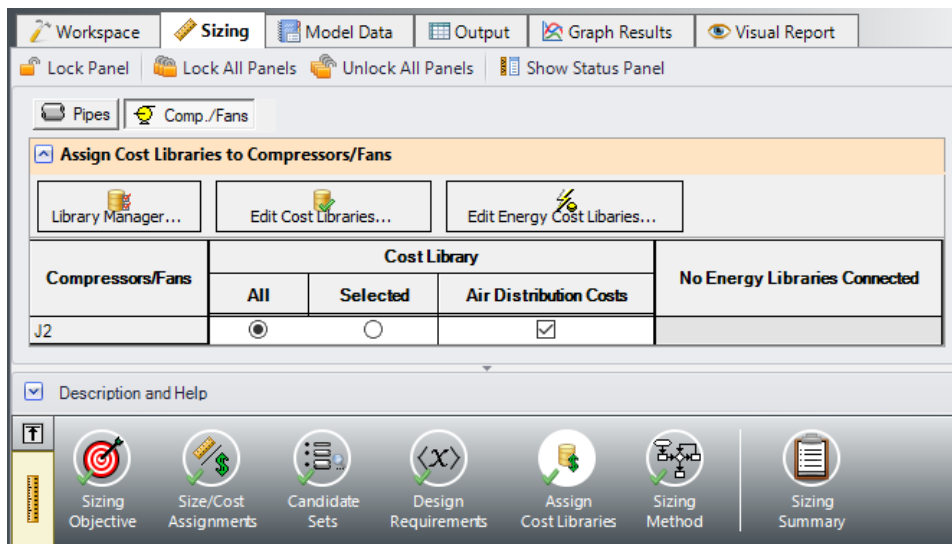


Figure 11: Assign Cost Libraries panel for compressors

## 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 five independent pipe sizes as noted on the Size/Cost Assignments panel (four groups and pipe P3), and two design requirements (one pipe design requirement applied to all of the pipes, and a minimum pressure drop applied to four control valves). Due to the small number of independent sizes, it would be recommended to try the MMFD or SQP search method. Though it is not shown here, if both methods are run for this scenario the same solution will be obtained, though the Sequential Quadratic Programming (SQP) Method is slightly faster.

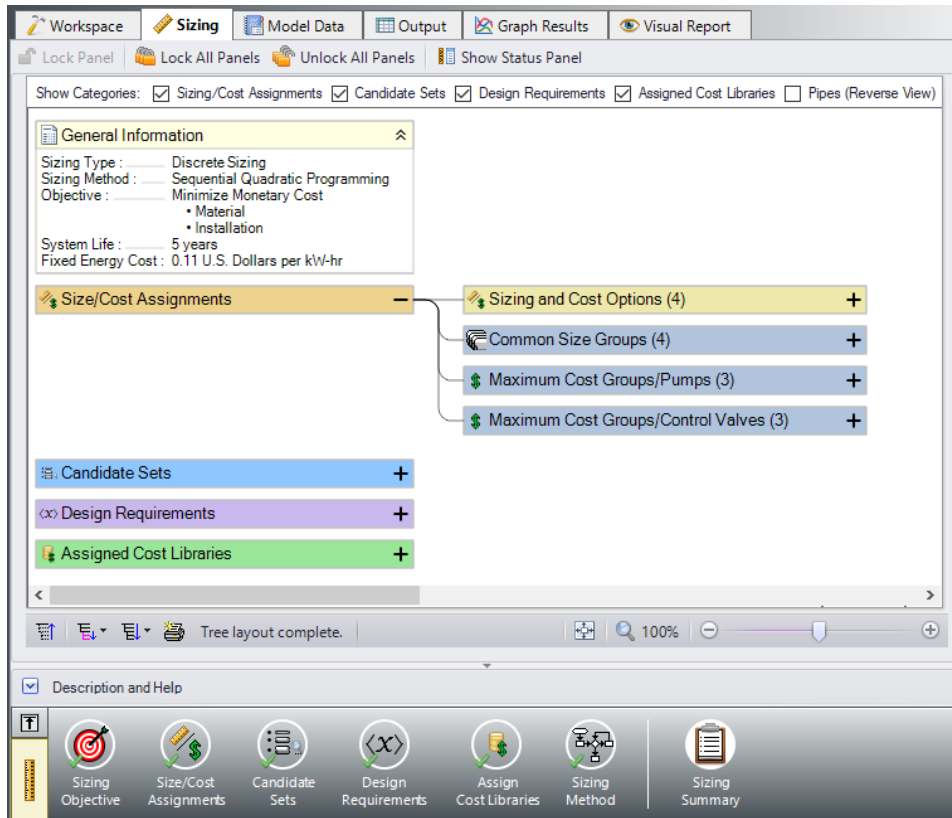
➤ **Select Sequential Quadratic Programming (SQP)** for the sizing method.

## 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 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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

## Step 5. Examine the Output

The Cost Report is shown in the General Section of the Output window (see Figure 13). The ANS module shows all costs in the Cost Report, including those that were not used in the automated sizing. The total cost for this system is \$198,098. This includes all costs over five years. The initial cost, which was the basis for the automated sizing, was the total of the material and installation (\$69,833). This total is listed as the total for the Items in Sizing line, 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 \$128,265. Looking across at the subtotal one can see that all of this cost is in energy (i.e., compressor power costs). The total cost is the sum of the two. For the next scenario this cost will be included in the automated sizing as a comparison.

<span>General</span> <span>Warnings</span> <span>Design Alerts</span> <span>Cost Report</span> <span>Compressor/Fan Summary</span> <span>Valve Summary</span> <span>Tank Sur</span>						
Table Units: U.S. Dollars	Material	Installation	Non-Recurring Sub Total	Operation/Energy	TOTAL	
<b>TOTAL OF ALL MODEL COSTS</b>					<b>198,098</b>	
<b>Total of All Shown Costs</b>	<b>42,422</b>	<b>27,411</b>	<b>69,833</b>	<b>128,265</b>	<b>198,098</b>	
<b>Items In Sizing</b>	<b>42,422</b>	<b>27,411</b>	<b>69,833</b>	<b>0</b>	<b>69,833</b>	
<b>Items Not In Sizing</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>128,265</b>	<b>128,265</b>	
<b>Pipe Subtotal</b>	<b>37,350</b>	<b>24,875</b>	<b>62,225</b>	<b>0</b>	<b>62,225</b>	
o P1	1,340	860	2,200	0	2,200	
o P2	3,350	2,150	5,500	0	5,500	
o P3	9,050	6,000	15,050	0	15,050	
o P4	4,525	3,000	7,525	0	7,525	
o P5	4,525	3,000	7,525	0	7,525	
o P6	4,525	3,000	7,525	0	7,525	
o P7	4,525	3,000	7,525	0	7,525	
o P8	4,525	3,000	7,525	0	7,525	
o P9	99	87	185	0	185	
o P10	99	87	185	0	185	
o P11	99	87	185	0	185	
o P12	99	87	185	0	185	
o P13	99	87	185	0	185	
o P14	99	87	185	0	185	
o P15	99	87	185	0	185	
o P16	99	87	185	0	185	
o P17	99	87	185	0	185	
o P18	99	87	185	0	185	
<b>Compressor/Fan Subtotal</b>	<b>5,072</b>	<b>2,536</b>	<b>7,608</b>	<b>128,265</b>	<b>135,873</b>	
o J2	5,072	2,536	7,608	128,265	135,873	

**Figure 13: The Cost Report in the Output window shows the total and individual costs (in thousands of 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 2-1/2 to 6 inches, as seen in Figure 14. The information for the compressors can be seen in the Compressor/Fan Summary tab in the General Section, which is also shown in Figure 14.

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 compressor came from the Air Distribution Costs library. If multiple 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 15.

# Air Distribution - ANS

Compressor/Fan Summary														
Jct	Results Diagram	Name	Mass Flow (lbm/sec)	Vol. Flow (ft3/sec)	dP Stag. (psid)	dP Static (psid)	dH Static (feet)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (hp)	Comp. Ratio Stag. (Percent)	BEP Q (ft3/sec)	BEP Mass (lbm/sec)	% of BEP (Percent)
2	Show	Compressor/Fan	1.591	21.51	5.787	5.807	9.977	80.00	N/A	35.72	139.7	N/A	N/A	N/A

Sizing					
Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (inches)
1	Pipe	Steel - ANSI	6 inch	STD (schedule 40)	6.065
2	Pipe	Steel - ANSI	6 inch	STD (schedule 40)	6.065
3	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
4	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
5	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
6	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
7	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
8	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
9	Pipe	Steel - ANSI	2-1/2 inch	STD (schedule 40)	2.469
10	Pipe	Steel - ANSI	2-1/2 inch	STD (schedule 40)	2.469
11	Pipe	Steel - ANSI	2-1/2 inch	STD (schedule 40)	2.469
12	Pipe	Steel - ANSI	2-1/2 inch	STD (schedule 40)	2.469
13	Pipe	Steel - ANSI	2-1/2 inch	STD (schedule 40)	2.469
14	Pipe	Steel - ANSI	2-1/2 inch	STD (schedule 40)	2.469
15	Pipe	Steel - ANSI	2-1/2 inch	STD (schedule 40)	2.469
16	Pipe	Steel - ANSI	2-1/2 inch	STD (schedule 40)	2.469
17	Pipe	Steel - ANSI	2-1/2 inch	STD (schedule 40)	2.469
18	Pipe	Steel - ANSI	2-1/2 inch	STD (schedule 40)	2.469

**Figure 14: Compressor and Pipe Sizes after automated sizing is performed**

## Air Distribution - ANS

Library Sources				
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	

Library Sources						
Jct	Name	Type	Material	Installation	Maintenance	Operation/Energy
1	Atmospheric Intake	Tank				
2	Compressor/Fan	Compressor/Fan	Air Distribution Costs	Air Distribution Costs		Fixed Energy Cost
3	Branch	Branch				
4	Branch	Branch				
5	Branch	Branch				
6	Branch	Branch				
7	Branch	Branch				
8	Branch	Branch				
9	Branch	Branch				
10	Control Valve	Control Valve				
11	Control Valve	Control Valve				
12	Control Valve	Control Valve				
13	Control Valve	Control Valve				
14	Control Valve	Control Valve				
15	Assigned Pressure	Assigned Pressure				
16	Assigned Pressure	Assigned Pressure				
17	Assigned Pressure	Assigned Pressure				
18	Assigned Pressure	Assigned Pressure				
19	Assigned Pressure	Assigned Pressure				

**Figure 15: Library Sources for the pipes and compressor in the Cost Report**

On 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 \$110,544 for this design, showing that the initial costs were reduced by about 37% from the initial design. We can also see that the initial design was feasible, meaning all of the design requirements were met (Figure 16).

Though not shown here, if the model is run using the objective to Calculate Costs, Do Not Size, it can be seen that the total cost for the system before sizing was \$178,626, which is actually less than the system that has been sized for initial costs. This is due to the fact that the larger pipe sizes in the initial system allowed for lower energy costs over the 5 year system life. This system is weighted towards energy costs, so any change in the energy costs will have a larger impact on the overall life cycle cost than the material and installation costs.

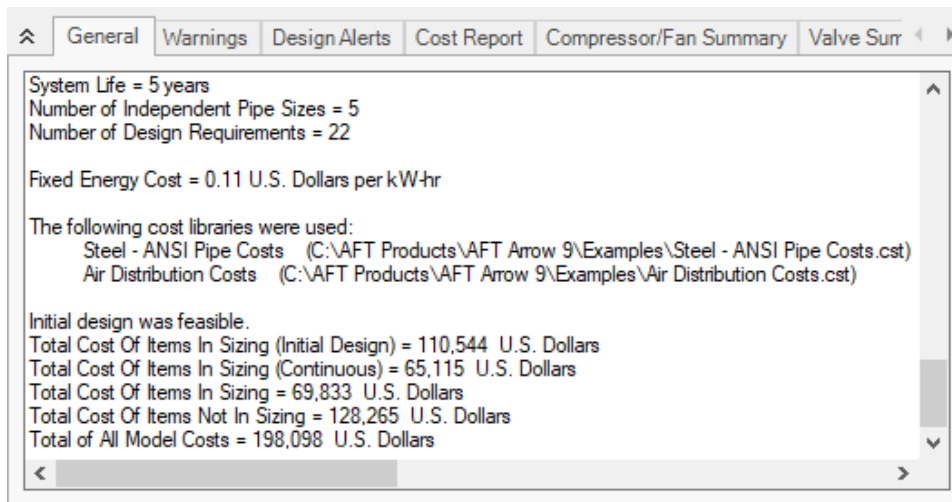


Figure 16: General section of Output window showing initial status of Items in Sizing

## Step 6. Size the System for Life Cycle Cost over Five Years

From the Scenario Manager in the Quick Access Panel, we will create two children scenarios. With the base scenario selected, right click and choose Create Child. Name the new scenario *Initial Cost*. This scenario will store the initial cost sizing results we just obtained. Repeat this process to create a second child scenario, and name it *Life Cycle Cost*. For this scenario the only change that will need to be made is to update the Sizing Objective.

Go to the Sizing window, and make sure that the Sizing Objective button is selected. 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 7. Examine the Output

After viewing the Cost Report, it can be seen that the overall cost is now \$166,167. The Items Not in Sizing line now has a value of zero, 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 17.

Taken on its own, this new cost represents a savings of about \$32,000 over the Initial cost scenario which was sized based on Initial Cost only. This represents a 14% cost reduction. In the initial cost scenario, the non-recurring cost was \$69,833, while the overall cost was \$198,098. Now the non-recurring cost is \$92,850 while the overall cost is \$166,167. The initial cost thus increased by about \$23,000 in order to reduce the operating cost from \$128,265 to \$73,317 (a reduction of about \$55,000). The source of the operating cost is the cost of power for the compressors. To reduce compressor power usage, it makes sense to increase the pipe size and thus reduce frictional losses. The larger pipe sizes can be reviewed by looking at the Sizing tab in the Pipe Output section, as shown in Figure 18.

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 \$69,833. Since the life cycle cost scenario total cost is 2.5 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.

<span>General</span> <span>Warnings</span> <span>Design Alerts</span> <span>Cost Report</span> <span>Compressor/Fan Summary</span> <span>Valve Summary</span> <span>Tank Sur</span>						
Table Units: U.S. Dollars	Material	Installation	Non-Recurring Sub Total	Operation/Energy	TOTAL	
<b>TOTAL OF ALL MODEL COSTS</b>					<b>166,167</b>	
<b>Total of All Shown Costs</b>	<b>57,492</b>	<b>35,358</b>	<b>92,850</b>	<b>73,317</b>	<b>166,167</b>	
<b>Items In Sizing</b>	<b>57,492</b>	<b>35,358</b>	<b>92,850</b>	<b>73,317</b>	<b>166,167</b>	
<b>Items Not In Sizing</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	
<b>Pipe Subtotal</b>	<b>54,450</b>	<b>33,838</b>	<b>88,288</b>	<b>0</b>	<b>88,288</b>	
o P1	2,040	1,000	3,040	0	3,040	
o P2	5,100	2,500	7,600	0	7,600	
o P3	13,000	8,325	21,325	0	21,325	
o P4	6,500	4,163	10,663	0	10,663	
o P5	6,500	4,163	10,663	0	10,663	
o P6	6,500	4,163	10,663	0	10,663	
o P7	6,500	4,163	10,663	0	10,663	
o P8	6,500	4,163	10,663	0	10,663	
o P9	181	120	301	0	301	
o P10	181	120	301	0	301	
o P11	181	120	301	0	301	
o P12	181	120	301	0	301	
o P13	181	120	301	0	301	
o P14	181	120	301	0	301	
o P15	181	120	301	0	301	
o P16	181	120	301	0	301	
o P17	181	120	301	0	301	
o P18	181	120	301	0	301	
<b>Compressor/Fan Subtotal</b>	<b>3,042</b>	<b>1,521</b>	<b>4,563</b>	<b>73,317</b>	<b>77,879</b>	
o J2	3,042	1,521	4,563	73,317	77,879	

Figure 17: Cost Report for the network after sizing for life cycle cost

Jct	Results Diagram	Name	Mass Flow (lbm/sec)	Vol. Flow (ft <sup>3</sup> /sec)	dP Stag (psid)	dP Static (psid)	dH Static (feet)	Overall Efficiency (Percent)	Speed (Percent)	Overall Power (hp)	Comp. Ratio Stag. (Percent)	BEP Q (ft <sup>3</sup> /sec)	BEP Mass (lbm/sec)	% of BEP
2	Show	Compressor/Fan	1.591	21.31	3.158	3.162	5.678	80.00	N/A	20.42	121.5	N/A	N/A	N/A

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (inches)
1	Pipe	Steel - ANSI	8 inch	STD (schedule 40)	7.981
2	Pipe	Steel - ANSI	8 inch	STD (schedule 40)	7.981
3	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
4	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
5	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
6	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
7	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
8	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
9	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
10	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
11	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
12	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
13	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
14	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
15	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
16	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
17	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
18	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026

Figure 18: Compressor and Pipe Sizes after automated sizing is performed for life cycle cost

## Step 8. Size the 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.

Clone the Life Cycle Cost scenario. Name the scenario *Initial Cost Limit*.

➤ **Return to the Sizing Objective panel** and select the option for Initial Cost Limit. We will set the limit as \$82,500 which is approximately an initial cost reduction of 25% from the original (un-sized) design. 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 five year life cycle can be seen in Table 1. Notice that while the material and installation costs were similar to the costs from the initial cost scenario, the energy costs were decreased by about \$10,000. These energy costs were achieved by increasing the compressor pipe sizes, while decreasing the header pipe sizes to achieve a better efficiency in the compressor. Table 2 shows a comparison of the pipe sizes throughout the system for the original design, and each of the sizing scenarios.

**Table 1: Cost Summary of Sizing Runs for the Air Distribution System**

Scenario	Material	Installation	Non-Recurring Sub Total	Energy	Total	Reduction
Not Sized	67,456	43,088	110,544	68,083	178,626	
Initial Cost	42,422	27,411	69,833	128,265	198,098	-20,000
Life Cycle Cost	57,492	35,358	92,850	73,317	166,167	12,600
Initial Cost Limit	45,138	27,971	73,109	117,888	190,997	-12,200

**Table 2: Summary of final pipe sizes for Common Size Groups in the Air Distribution System**

Scenario	Compressor	Header A	Header B	Discharge	Pipe P3
Not Sized	6 inch	6 inch	6 inch	3 inch	6 inch
Initial Cost	6 inch	4 inch	4 inch	2-1/2 inch	4 inch
Life Cycle Cost	8 inch	5 inch	5 inch	4 inch	5 inch
Initial Cost Limit	8 inch	4 inch	4 inch	3-1/2 inch	4 inch

### Step 9. Automated Sizing with Compressor Curve Data

As discussed in the previous section summarizing the compressor selection process, once the compressor is sized then actual compressors can be modeled. The actual compressor should closely match the sizing results in the following areas: generated pressure at the design flow, efficiency at the design flow, and cost. Let's apply actual compressor data for the case that was sized for life cycle cost with an initial cost limit to complete phase two of the compressor selection process above.

Reviewing the results for the life cycle cost with initial cost limit scenario one can see that the sized system calls for a compressor of about 33 hp that generates about 5.3 psid of pressure at 1250 scfm. The nominal efficiency used in the sizing part of the analysis was 80%. The material cost for such a compressor was about \$4,800, and the installation cost was about \$2,400. Note that if no actual compressor can be found that reflects these requirements, then the phase 1 of the compressor selection process described above should be repeated with better performance and/or cost data for the compressor.

Based on the sizing analysis, we will choose a compressor from the (fictitious) Rocky Mountain Compressor Company. Its model 100XLC is a close match for the requirements of a 5-year life cycle design. The model 100XLC has the following characteristics: at 1250 scfm its pressure is 5.6 psid and efficiency is 80%. Its material cost is \$5,800 and installation cost is \$2,400. To setup a new scenario for the system with the candidate compressor data:

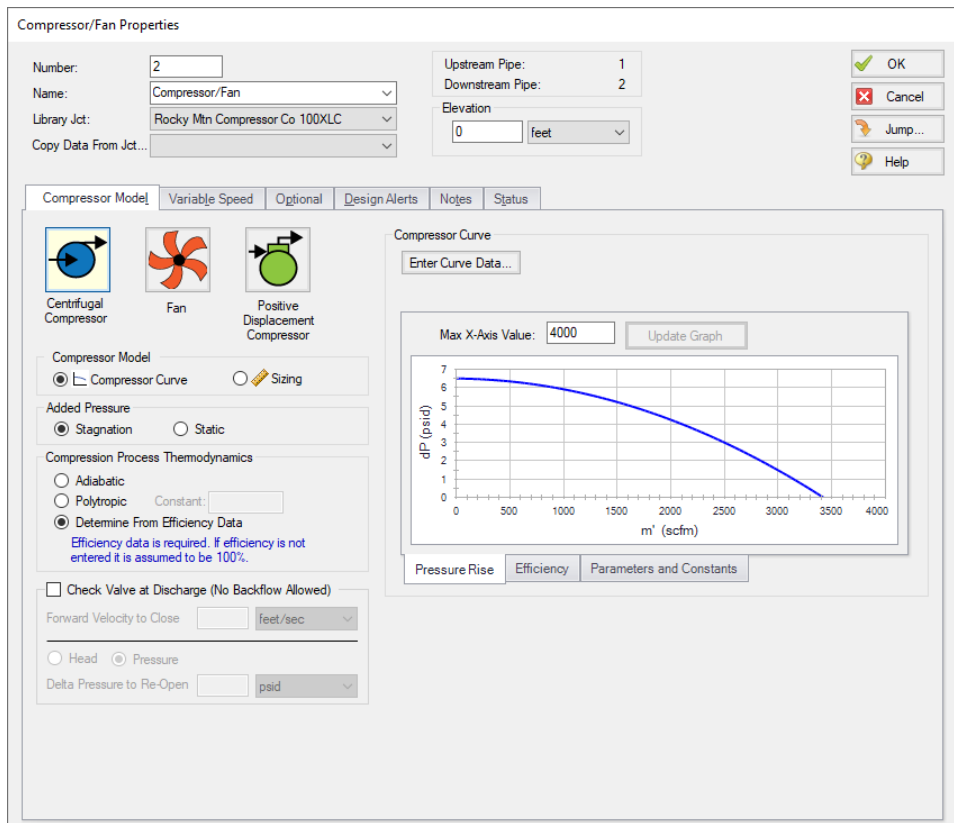
In the Scenario Manager clone the Initial Cost Limit child scenario and name it *Actual Compressor Curve*.

In the new scenario, open the properties window for compressor J2 select Rocky Mtn Compressor Co 100XLC from the Library Jct list to import the compressor data from the library as shown in Figure 19, then click OK.

In the Flow Control Valve Property window for J12, change the Valve Type to Flow Control (FCV), and enter a Flow Setpoint of 250 scfm. Now that we have changed the control valve model, we will need to apply a Design Requirement in order to account for the minimum pressure drop.

Go to the Sizing window, then navigate to the Design Requirements panel. Select the Control Valves button at the top of the window.

Apply the Min Pressure Drop design requirement to Control Valve J12.



**Figure 19: Compressor Properties window with the Rocky Mtn Compressor Co 100XLC library item selected**

The necessary changes are now complete to size the system with the updated compressor and flow control valve data.

➤ **Select Run from the Analysis menu** to perform the automated sizing. When the solver has finished, click Output to review the results, as can be seen in Figure 20.

The Cost Report shows a total sized cost of \$197,225. Comparing the final pipe sizes to the Initial Cost Limit scenario shows the pipe sizes and compressor operating point are similar.

Now we have sized the system for use with an actual compressor.



Air Distribution - ANS

<span>General</span> <span>Warnings</span> <span>Design Alerts</span> <span>Cost Report</span> <span>Compressor/Fan Summary</span> <span>Valve Summary</span> <span>Tank Sur</span>						
Table Units: U.S. Dollars	Material	Installation	Non-Recurring Sub Total	Operation/Energy	TOTAL	
<b>TOTAL OF ALL MODEL COSTS</b>						<b>197,225</b>
<b>Total of All Shown Costs</b>	<b>45,600</b>	<b>27,765</b>	<b>73,365</b>	<b>123,860</b>	<b>197,225</b>	
<b>Items In Sizing</b>	<b>45,600</b>	<b>27,765</b>	<b>73,365</b>	<b>123,860</b>	<b>197,225</b>	
<b>Items Not In Sizing</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	<b>0</b>	
<b>Pipe Subtotal</b>	<b>39,800</b>	<b>25,365</b>	<b>65,165</b>	<b>0</b>	<b>65,165</b>	
o P1	2,040	1,000	3,040	0	3,040	
o P2	5,100	2,500	7,600	0	7,600	
o P3	9,050	6,000	15,050	0	15,050	
o P4	4,525	3,000	7,525	0	7,525	
o P5	4,525	3,000	7,525	0	7,525	
o P6	4,525	3,000	7,525	0	7,525	
o P7	4,525	3,000	7,525	0	7,525	
o P8	4,525	3,000	7,525	0	7,525	
o P9	99	87	185	0	185	
o P10	99	87	185	0	185	
o P11	99	87	185	0	185	
o P12	99	87	185	0	185	
o P13	99	87	185	0	185	
o P14	99	87	185	0	185	
o P15	99	87	185	0	185	
o P16	99	87	185	0	185	
o P17	99	87	185	0	185	
o P18	99	87	185	0	185	
<b>Compressor/Fan Subtotal</b>	<b>5,800</b>	<b>2,400</b>	<b>8,200</b>	<b>123,860</b>	<b>132,060</b>	
o J2	5,800	2,400	8,200	123,860	132,060	

**Figure 20: Cost Report for system with actual compressor and FCV data sized for five year life cycle cost with initial cost limit**

# Air Distribution, Multiple Design Cases - ANS

## Summary

---

Some system designs have multiple operating conditions to satisfy. For instance, the air distribution system discussed in the previous example may have a requirement that it operate with one of the discharge valves closed. 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

- Using Dependent Design Cases
- Using Duplicate Special to create Dependent Designs
- Understanding fractional compressor costs with Dependent Design Cases
- Specifying Multiple Dependent Design Cases

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

In addition, it is assumed that the user has worked through the [Beginner: Three Tank Steam System - ANS](#) example, and is familiar with the basics of ANS analysis.

This example is an addendum to the [Concurrently Sizing a Compressor and 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 Arrow installation:

- ***US - Air Distribution - ANS Initial.aro***
- ***US - Air Distribution - ANS Final.aro***
- ***Air Distribution.dat*** - engineering library
- ***Air Distribution Costs.cst*** - cost library for ***Air Distribution.dat***
- ***Steel - ANSI Pipe Cost.cst*** - cost library for ***Steel - ANSI pipes***

## Step 1. Start AFT Arrow

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

Open the **US - Air Distribution - ANS Initial.aro** example file listed above, which is located in the Examples folder in the AFT Arrow application folder. Save the file to a different folder. Complete the steps in the example [Concurrently Sizing a Compressor and System - ANS](#) until the Life Cycle Cost scenario is completed. Clone this scenario and name it *Dependent Design Case*.

Alternatively, open the **US - Air Distribution - ANS Final.aro** example file. Right-click the Life Cycle Cost scenario and select Save Scenario to File Without Children. Save the scenario to a new location then open this new model file. Create a child from the base scenario and name it *Dependent Design Case*.

## Step 2. Define the Pipes and Junctions Group

The Workspace should appear as shown in the figure below.

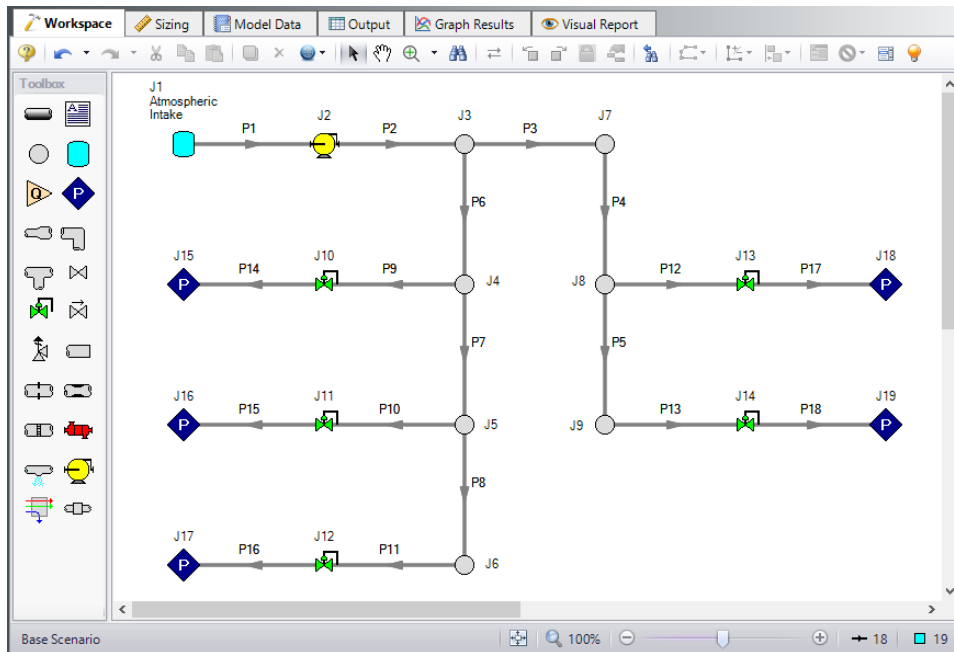


Figure 1: Workspace for Air Distribution model

### Step 3. Enable Dependent Design

---

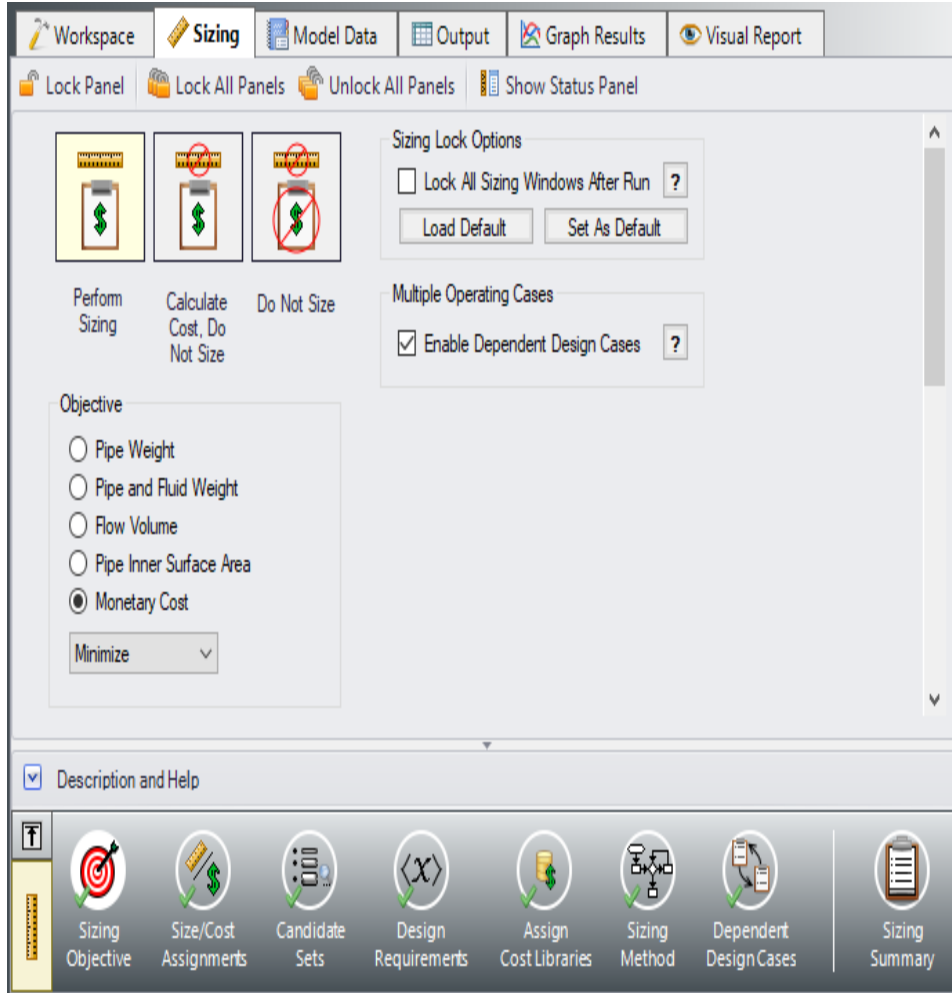
When modeling multiple design cases, the different cases need to be broken up into a primary design case, of which there is one, and dependent design cases, of which there can be any number.

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 discharge valves open. We want to also consider the case where one discharge is closed. The one discharge closed case will be created as a dependent design case.

To start creating the dependent design case, we will first need to enable the Dependent Design Case modeling:

➤ **Go to the Sizing Objective panel** and select the Enable Dependent Design Cases option. A new button will now be available for the Dependent Design Cases panel in the Sizing Navigation Panel, as can be seen in Figure 2.



*Figure 2: Dependent Design Cases are enabled from the Sizing Objective panel*

## Step 4. Set up the Dependent Design Case

### A. Use Duplicate Special

The easiest way to set up a dependent design case is to use Duplicate Special, which will allow us to make an exact duplicate of the model in order to set up the additional operating cases. Note that you should fully define the model and all of the sizing settings for the primary case before creating the dependent cases, as has been done in this model.

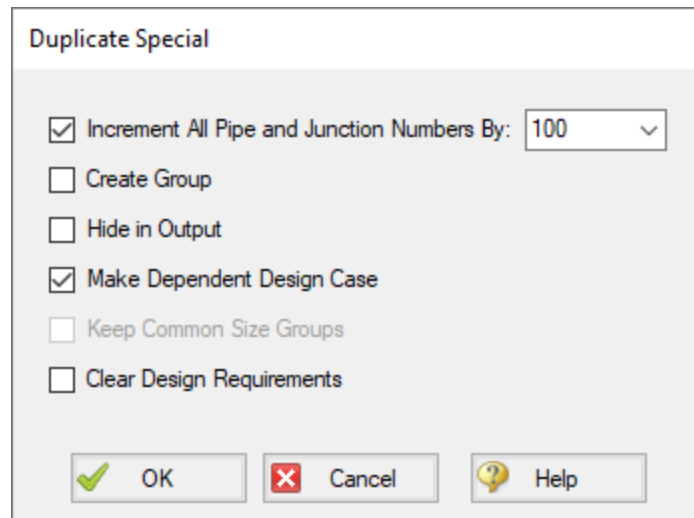
To create the dependent design case:

1. Make sure that the scenario where the dependent cases will be created is loaded. In this case, you should have the Dependent Design Case scenario loaded.
2. Go to the Workspace window.

3. Choose Select All from the Edit menu, or use CTRL+A.
4. Choose Duplicate Special from the Edit menu (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 Work-space. 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.
- 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**

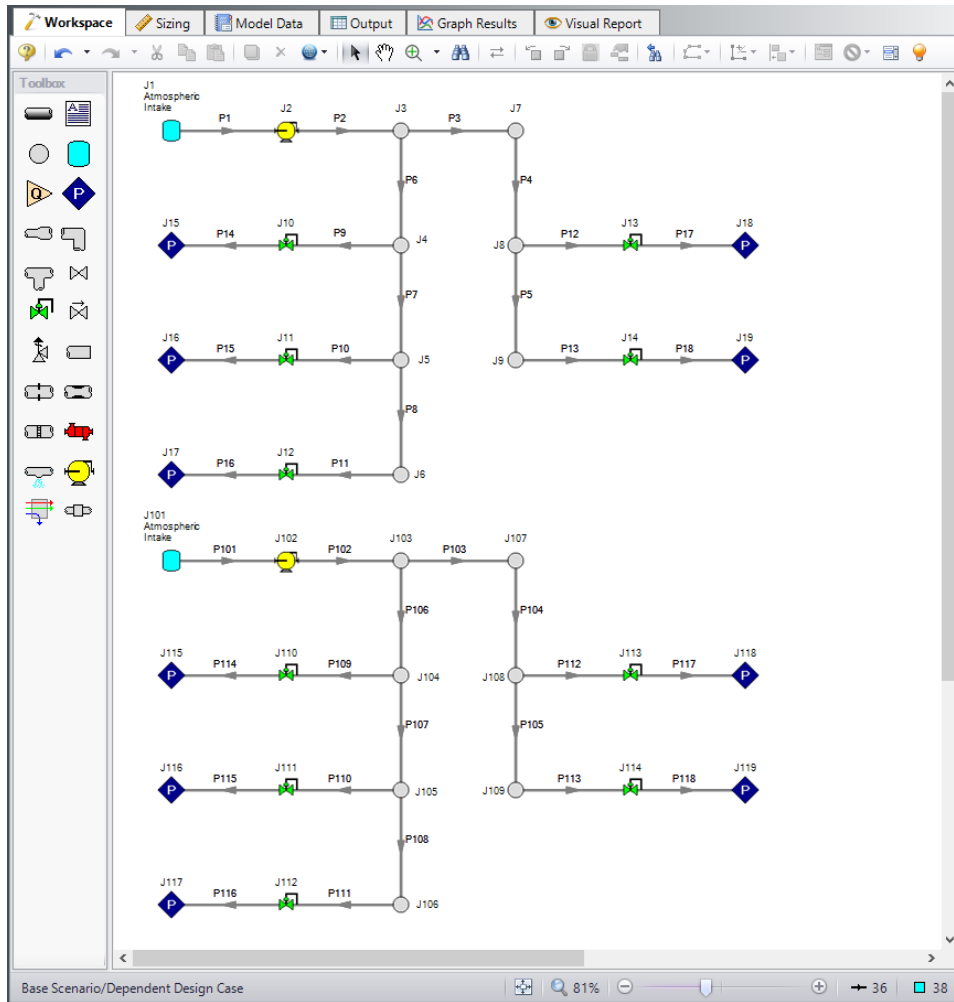



Figure 4: Workspace with the Dependent Design Case created for the Air Distribution model

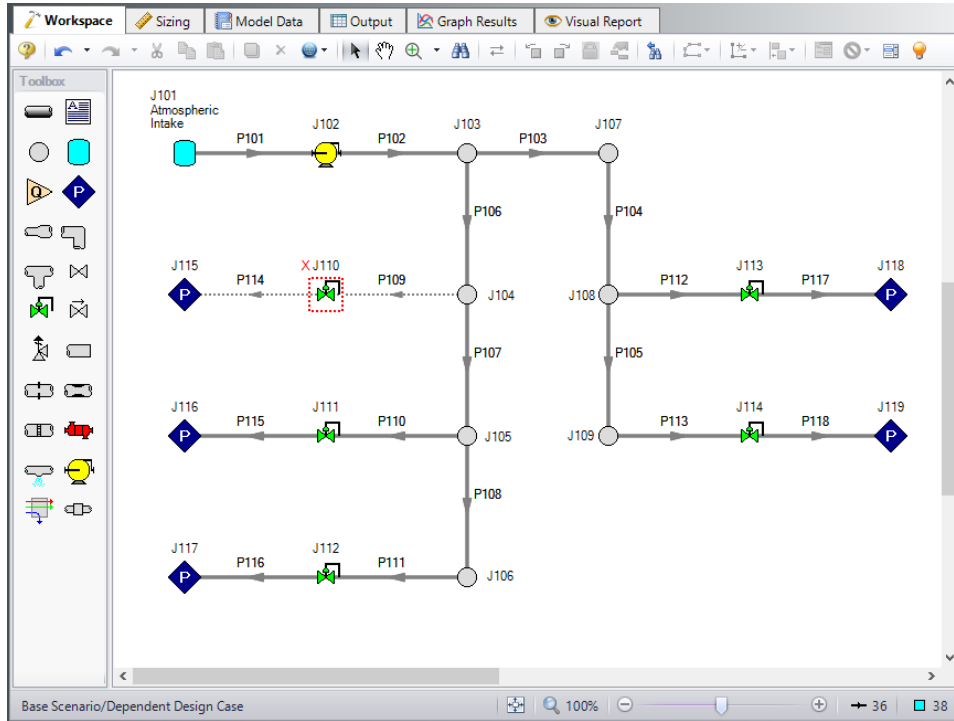
## B. Apply the Dependent Design Requirements

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 compressor'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 Discharge Room in the system is shut off, so we will use Special Conditions on the discharge valve to that room.

Select Control Valve J110 in the dependent case, and use the Special Conditions button  on the Edit menu to set the junction to Closed. The dependent case should now appear as shown in Figure 5.



**Figure 5: Flow off for Discharge Room A1 in the Dependent Design Case**

With one valve shutoff we will need to distribute the Fixed Mass Flow Rate of the compressor for the four Discharge Rooms remaining. That means the new Mass Flow Rate for every Control Valve will be 312.5 scfm. The flow requirements for the Control Valves J111, J113 and J114 need to be changed from 250 scfm to 312.5 scfm. Note that Control Valve J112 will not need to be changed since it is set to a fixed pressure drop for sizing purposes, and still has the same pressure drop requirement.

➤ **Open the Control Valve Properties window** for control valve J111 and change the flow setpoint to 312.5 scfm, then repeat this for junctions J113 and J114 as discussed above.

Since we have closed control valve J110 it is necessary to remove the minimum pressure drop requirement from this valve so that the results will not be skewed.

➤ **Go to the Design Requirements panel** in the Sizing window and select the Control Valves button. Un-check the box for the pressure drop requirement next to J110 to remove it from the closed valve as shown in Figure 6.

For this scenario we also want to change the Sizing Method, since the Modified Method of Feasible Directions was determined to be the most efficient for this model.

➤ **Click the Sizing Method panel button** in the Sizing Navigation panel, then select the Modified Method of Feasible Directions (MMFD) from the Search Method list.



The Dependent Design case is now complete, since it has inherited the rest of its sizing settings from the primary case. Select Run from the Analysis menu and got to the Output window to view the results once the run is complete.

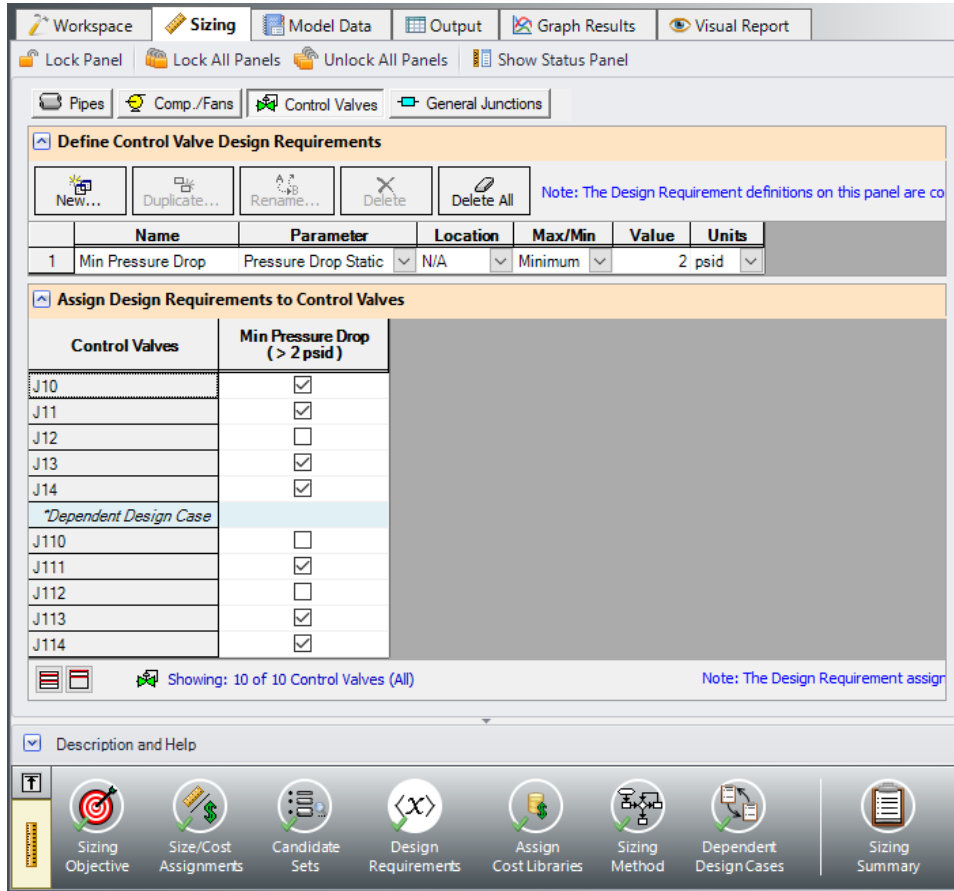


Figure 6: Control Valve Design Requirements adjusted for the Dependent Design Case

## Summary

The following summary Table 1 was compiled for the two scenarios. It can be observed that when the dependent design case is included in the sizing, the cost increases by approximately 3%. 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. In this case the pipe sizes needed to be increased to account for the additional load when one discharge valve is closed. The pipe sizes for the Dependent Design Case scenario can be seen in Figure 7.

**Table 1: Summary of Sizing Runs for Air Distribution system**

Scenario	Life Cycle Cost	Dependent Design Case
Meets all Requirements	Yes	Yes
Material	\$57,492	\$61,242
Installation	\$35,358	\$37,783
Operation	\$73,317	\$73,317
Total	\$166,167	\$172,342

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (inches)
1	Pipe	Steel - ANSI	8 inch	STD (schedule 40)	7.981
2	Pipe	Steel - ANSI	8 inch	STD (schedule 40)	7.981
3	Pipe	Steel - ANSI	6 inch	STD (schedule 40)	6.065
4	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
5	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
6	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
7	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
8	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
9	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
10	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
11	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
12	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
13	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
14	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
15	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
16	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
17	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
18	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026

**Figure 7: Pipe size results for the system with the dependent design case included**

# Helium Transfer - ANS

## Summary

---

This example will show how to use the ANS module to size a system based on pipe weight to deliver a minimum flow of helium for variable inlet temperature.

---

**Note:** This example can only be run if you have a license for the ANS module.

---

## Topics Covered

- Sizing using pipe weight
- Creating Common Size Groups
- Using dependent design cases to satisfy two different operating modes for a system

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

In addition, it is assumed that the user has worked through the [Beginner: Three Tank Steam System - 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 Arrow installation:

- *US - Helium Transfer - ANS Initial.aro*
- *US - Helium Transfer - ANS Final.aro*

## Step 1. Start AFT Arrow

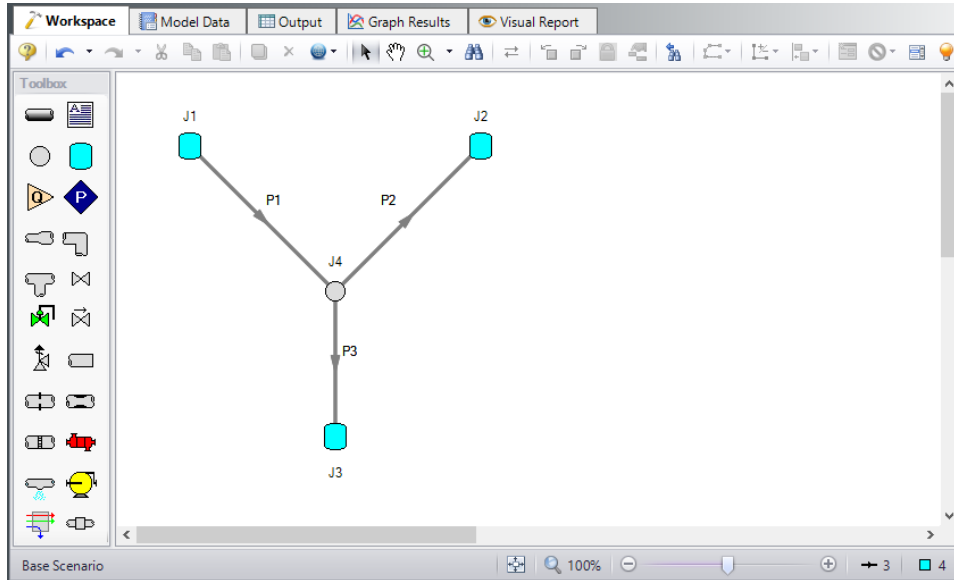
---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

Open the **US - Helium Transfer - ANS Initial.aro** example file listed above, which is located in the Examples folder in the AFT Arrow application folder. Save the file to a different folder. The Workspace should look like Figure 1 below.

In this system, Helium flows from one tank to two other delivery tanks. Throughout the year its inlet temperature varies from 32 to 100 deg. F. There is a minimum flowrate required to each tank of 2.5 lbm/sec. Since the supply temperature can vary we will analyze two different design cases, one for the maximum temperature (Hot Case), and one for the minimum temperature (Cold Case). We are using the Hot Case as our primary design, so the model inputs currently reflect the Hot Case. We will need to first define the sizing settings for the primary case, then set up the secondary case using the cold temperatures.



**Figure 1: Model layout for normal system operation**

## Step 2. Define the Modules Group

Navigate to the Modules panel in Analysis Setup. Check the box next to Activate ANS. The Network option should automatically be selected, making ANS enabled for use.

## Step 3. Define the Automatic Sizing Group

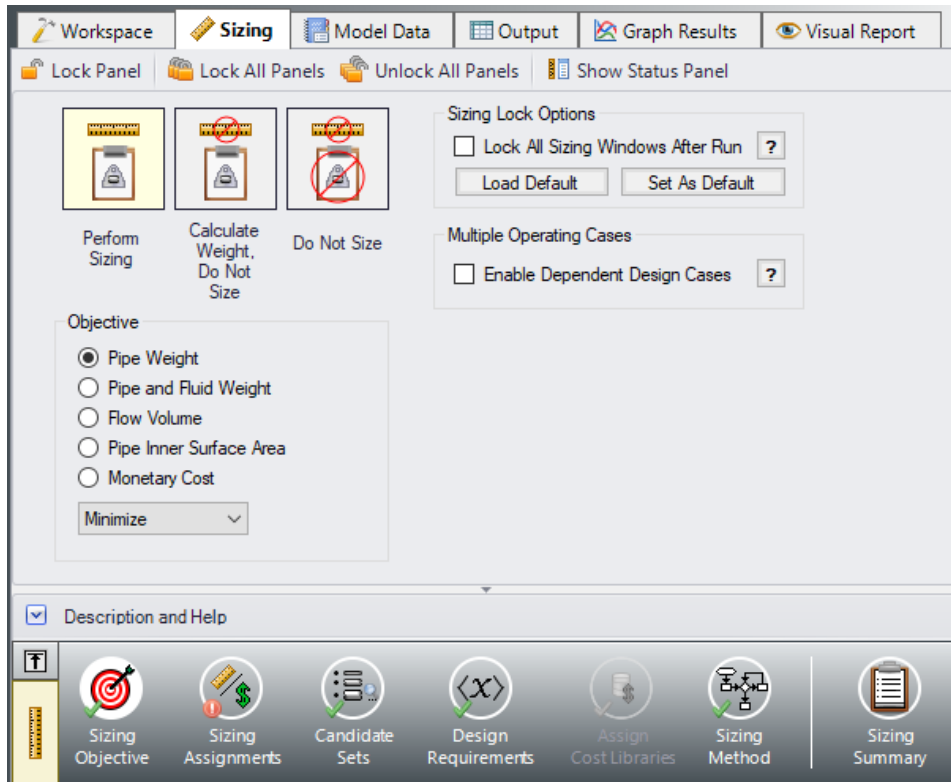
To define the Automatic Sizing group, go to the Sizing window by clicking on the Sizing tab.

### A. Sizing Objective

Go to the Sizing window by selecting the Sizing tab. 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. Select Perform Sizing for this calculation.
2. For the Objective, choose Pipe Weight, and select the Minimize option from the drop-down list if it is not already selected.

The Sizing Objective window should now appear as shown in Figure 2.



**Figure 2: Sizing Objective panel setup to minimize pipe weight**

## B. Sizing Assignments

On the Sizing Navigation panel select the Sizing Assignments button.

For this system we will be sizing all of the pipes since it is a new system.

➤ **Move each of the pipes** to Always Include in Weight.

The piping to each of the discharge tanks in the system will be identical, so a Common Size Group will need to be created to link the sizes in the two pipes.

Above the Pipe Grouping table select New, then give the group the name *Discharge Pipes*. Select the radio buttons to place pipes P2 and P3 in the Common Size Group.

The completed pipe Sizing Assignments can be seen in Figure 3.

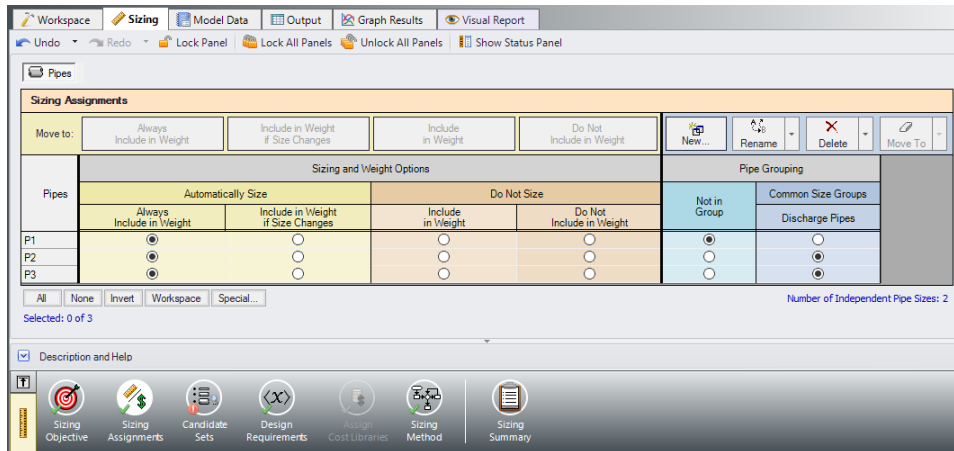


Figure 3: Sizing Assignments fully defined for the model

### C. Candidate Sets

Click on the Candidate Sets button to open the Candidate Sets panel.

For this system it is desired to use STD Steel - ANSI. To create a Candidate Set, do the following:

1. Under Define Candidate Sets, click New.
2. Give the set the name *STD Pipes* and click OK.
3. From the drop down list choose Steel - ANSI.
4. In the Available Material Sizes and Types on the left, expand the STD pipe sizes list.
5. Double-click each of the sizes from 1 to 10 inches to add them to the list on the right.
6. In the Select Pipe Sizes window, click OK.

We now need to define which pipes will use this Candidate Set during the sizing calculation. Select the radio buttons next to pipe P1 and the Common Size Group to assign the Candidate Set to each of the pipes, as shown in Figure 4.

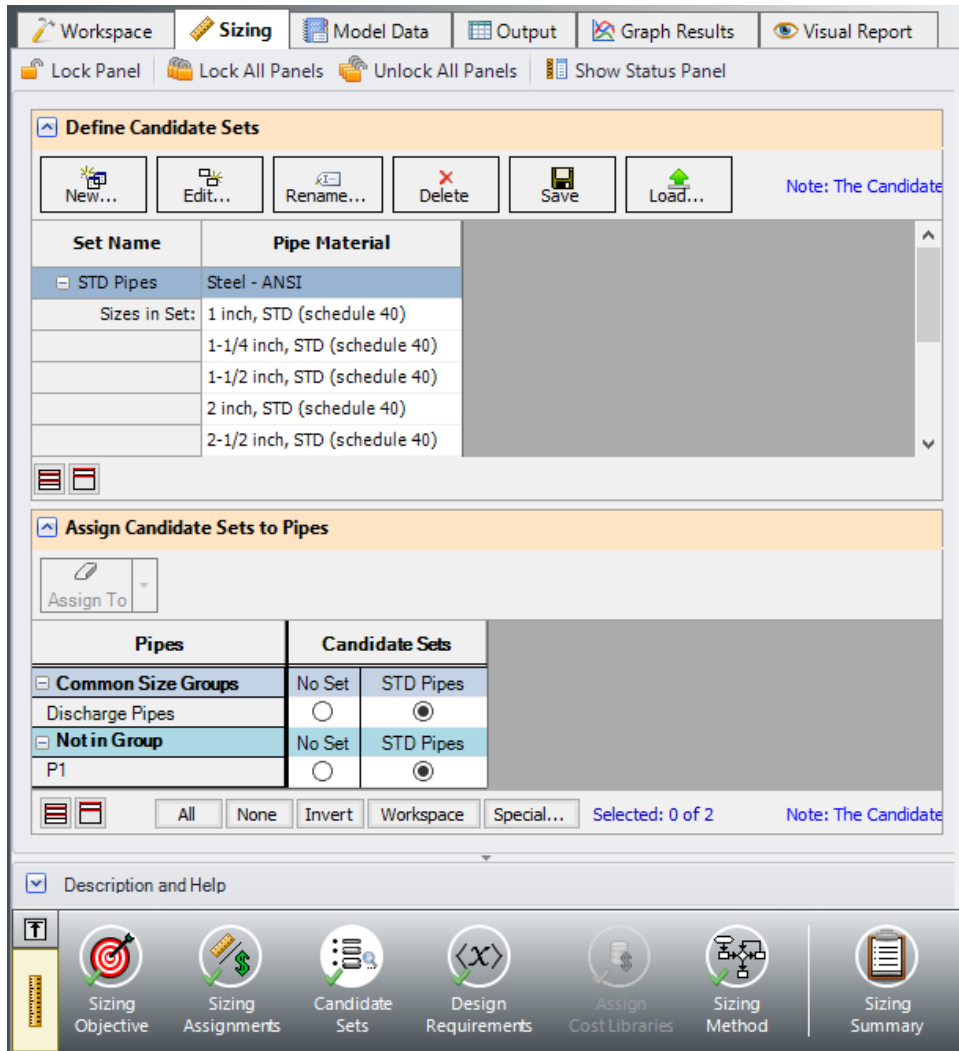


Figure 4: Candidate Sets panel fully defined for the model

## D. Design Requirements

Select the Design Requirements button from the Navigation Panel.

For this system there is only one Design Requirement, which is the requirement to maintain a minimum flow of 2.5 lbm/sec into each of the discharge tanks. To define this requirement:

1. Click New under Define Pipe Design Requirements.
2. Enter the name *Minimum Flow*.
3. Select Mass Flow Rate as the Parameter.
4. Choose Minimum for Max/Min, and enter 2.5 lbm/sec.

Now we need to apply the defined Design Requirement to the discharge pipes.

➤ **Check the box next to Discharge Pipes** in the Assign Design Requirements to Pipes section. The Design Requirements panel should now appear as shown in Figure 5.

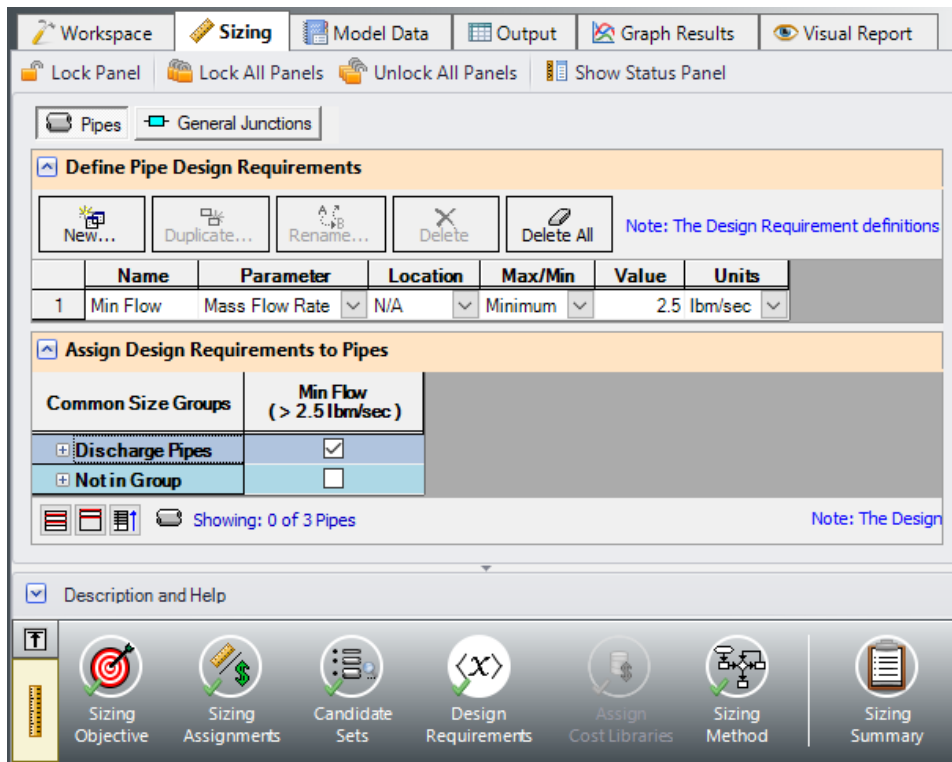


Figure 5: Design Requirements panel for pipes

## 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 disabled.

## 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.

This model is relatively simple, so the default, Modified Method of Feasible Directions (MMFD), should be sufficient. Ensure that this method is selected from the list of Search Methods.

## G. Dependent Design Cases



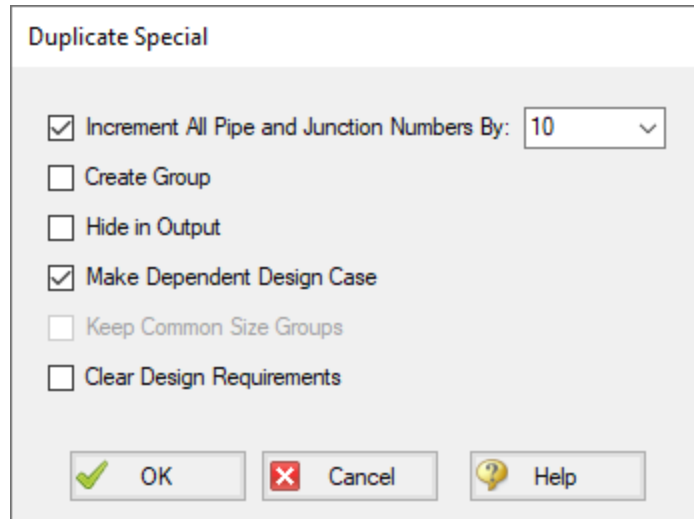
We have now set up the system to be sized for the primary design case, but we will also need to account for the cold case where the system is operating at the minimum inlet temperature. To do this we will create a dependent design case by completing the following steps:

1. Go to the Sizing Objective panel and select the Enable Dependent Design Cases option. 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 choose Select All from the Edit menu.
4. Open Duplicate Special (from the Edit menu), enter an increment of 10 and select Make Dependent Design Case (Figure 6). Click OK.
5. Move the duplicated pipes and junctions to distinguish them from the original ones from the Primary Design Case. The Workspace should now appear as shown in Figure 7.
6. In the Cold Case inlet tank J11, open the Tank Properties window and change the temperature to 32 deg. F.

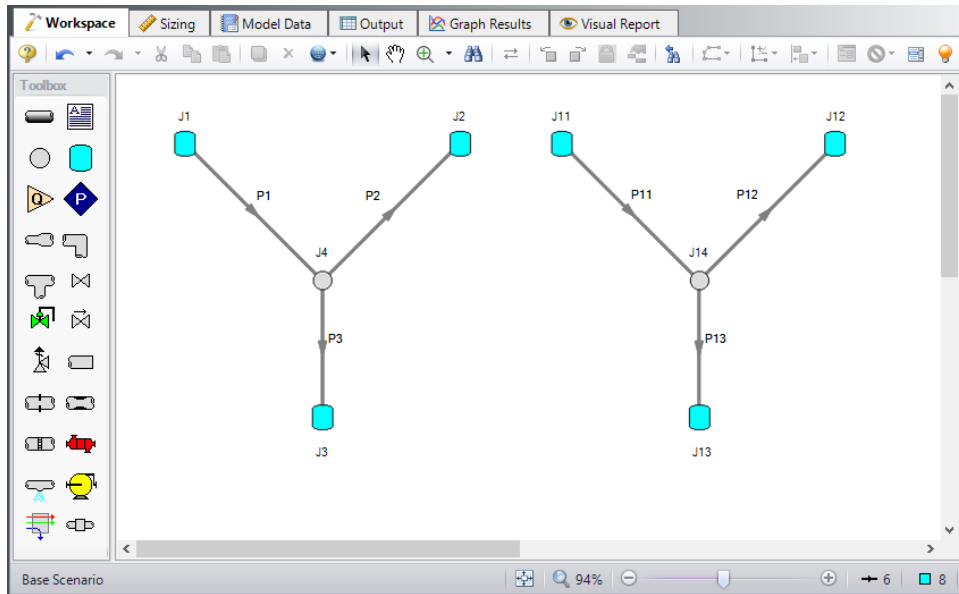
Return to the Dependent Design Cases panel.

When Duplicate Special was performed with Dependent Design Case selected, each of the duplicated pipes was created with a special type of grouping. For example, pipe P11 is grouped with pipe P1 as a DDC pipe (see Figure 8). 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.

It should also be noted that the dependent design grouping causes each of the dependent pipes to inherit their Common Size Groups, Candidate Sets, and Cost Library settings (if applicable) from the pipes in the primary case. The dependent design case pipes are therefore hidden on all sizing panels except for the Design Requirements and Dependent Design Cases panels.



**Figure 6: Duplicate Special settings to create a dependent design case**



**Figure 7: Workspace with dependent design case after duplication**

Pipes	DDC Pipe?	Assigned Design Requirements
P1	NO	None
P11	YES (P1)	None
P2	NO	Min Flow
P12	YES (P2)	Min Flow
P3	NO	Min Flow
P13	YES (P3)	Min Flow

**Figure 8: Pipes in Dependent Design Cases have a special grouping relationship with pipes in the primary case**

## 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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

## Step 5. Examine the Output

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 9. It can be seen that the 5 inch and 3 inch pipe sizes were chosen for the header and discharge pipes, respectively. The overall weight of the system is 22,194 lbm.

Table Units: lbm		Pipe Weight
<b>Total of All Shown Costs</b>		<b>22,194</b>
<b>Items In Sizing</b>		<b>22,194</b>
<b>Items Not In Sizing</b>		<b>0</b>
<b>Pipe Subtotal</b>		<b>22,194</b>
o P1		14,618
o P2		3,788
o P3		3,788

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (inches)
1	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
2	Pipe	Steel - ANSI	3 inch	STD (schedule 40)	3.068
3	Pipe	Steel - ANSI	3 inch	STD (schedule 40)	3.068
11	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
12	Pipe	Steel - ANSI	3 inch	STD (schedule 40)	3.068
13	Pipe	Steel - ANSI	3 inch	STD (schedule 40)	3.068

Figure 9: Final pipe sizes and Cost Report for the Helium Transfer system considering the hot and cold cases

## Conclusion

By creating a dependent design case both the hot and cold temperature operating cases were able to be taken into account in the automated sizing for this simple system. In more complex systems additional

dependent cases could be created to consider various system configurations as well as further operating conditions.

# Natural Gas Burner - ANS

## Summary

---

This example will size the pipes for a natural gas supply system to five burners. It is desired to analyze two separate design cases using monetary cost automated sizing.

---

**Note:** This example can only be run if you have a license for the ANS module.

---

## Topics Covered

- Sizing using monetary cost
- Using dependent design cases to satisfy two different operating modes for a system
- Understanding the effect of monetary vs. non-monetary sizing objectives

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

In addition, it is assumed that the user has worked through the [Beginner: Three Tank Steam System - 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 Arrow installation:

- ***US - Natural Gas Burner - ANS Initial.aro***
- ***US - Natural Gas Burner - ANS Final.aro***
- ***Steel - ANSI Pipe Costs.cst*** - cost library for **Steel - ANSI** pipes

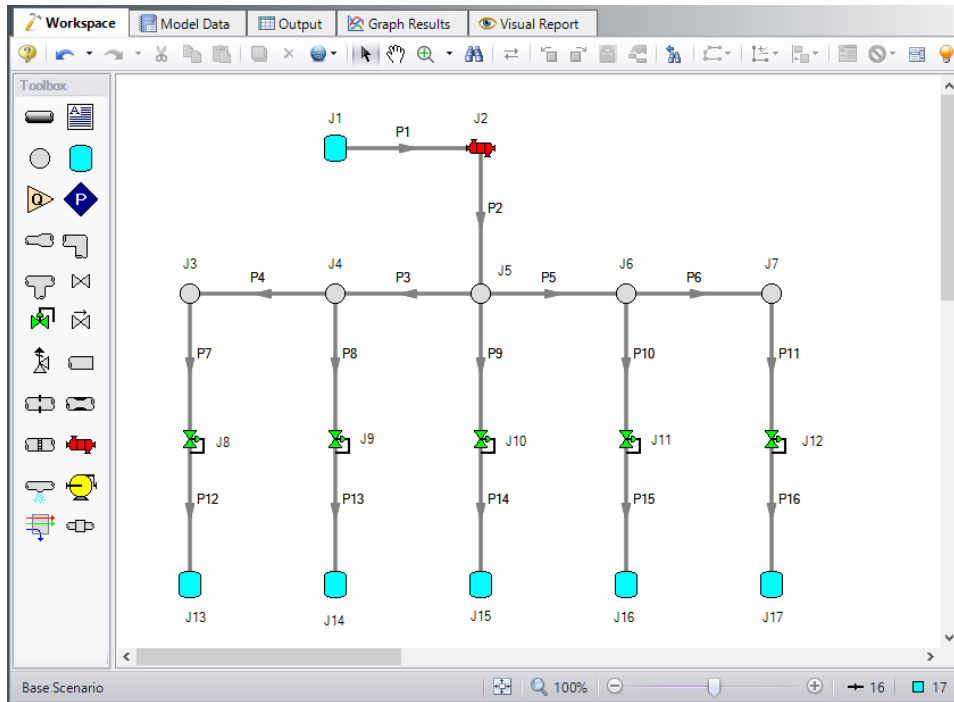
## Step 1. Start AFT Arrow

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

Open the **US - Natural Gas Burner - ANS Initial.aro** example file listed above, which is located in the Examples folder in the AFT Arrow application folder. Save the file to a different folder. The Workspace should look like Figure 1 below.



**Figure 1: Natural Gas Burner model layout in Workspace**

## Step 2. Define the Modules Group

Navigate to the Modules panel in Analysis Setup. Check the box next to Activate ANS. The Network option should automatically be selected, making ANS enabled for use.

## Step 3. Define the Automatic Sizing Group

For this system we will be sizing the natural gas system for two different operating cases, one with all burners running and one with the third burner closed. 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 case with all burners on will be used as the primary design case.

To start we will set up the sizing settings for the primary design case, after which we will set up the dependent design case.

### **A. Sizing Objective**

The Sizing Objective panel should be selected by default. For this analysis, we are interested in sizing considering the monetary cost for the initial costs only.

1. Set the Sizing Option to Perform Sizing.
2. For the Objective, choose Monetary Cost, and select the Minimize option from the drop-down list.
3. 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.

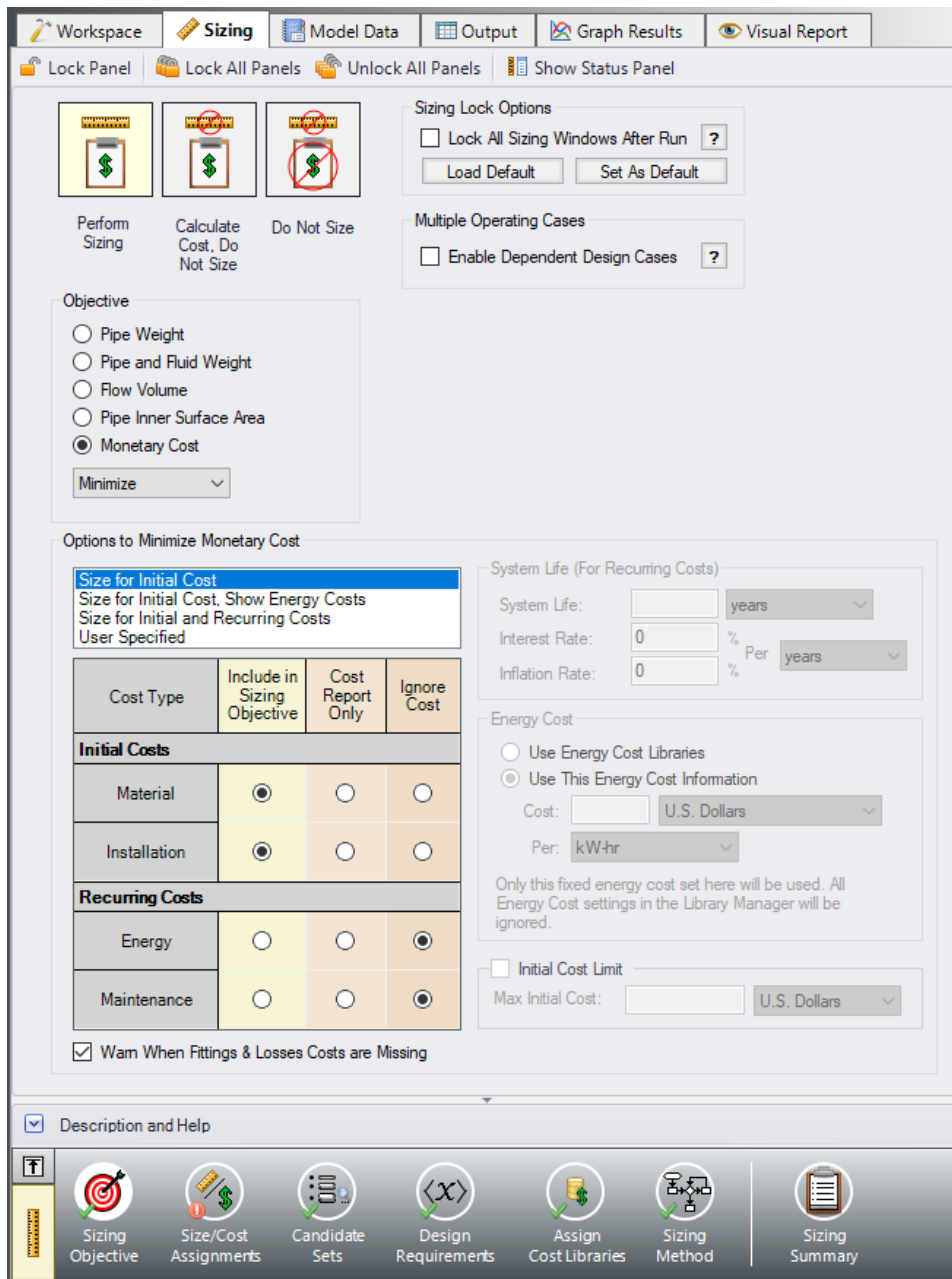


Figure 2: Sizing Objective window setup for an Initial Cost analysis

## B. Size/Cost Assignments

On the toolbar at the bottom of the window select the Size/Cost Assignments button.

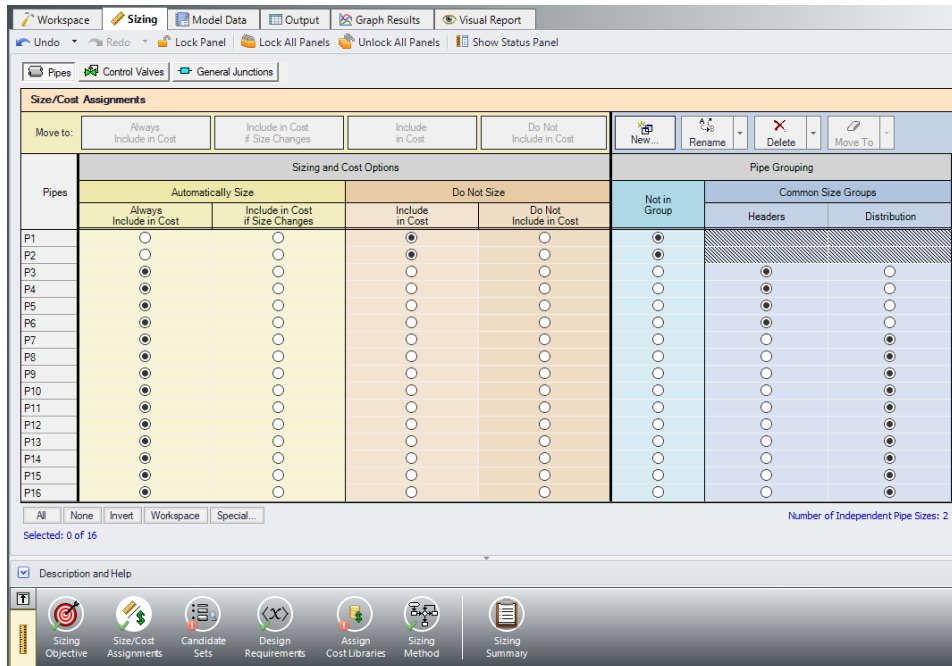
For the supply lines P1 and P2, the pipe size is fixed to 6 inches and will not be sized. However, we will still need to purchase these pipes, so we will consider their cost. The rest of the pipes in the system will need to be sized for the design.



The 4 header pipes (P3, P4, P5, P6) represent a single header in the physical system, so it will be necessary to add them to a Common Size Group. A second Common Size Group should be created to contain all of the discharge piping (P7 - P16), as each of the branches to the burners should be identical. To set up the Size/Cost Assignments as described, do the following:

1. Move pipes P1 & P2 to Do Not Size – Include in Cost in the table. This will calculate their cost while keeping their size fixed.
2. Select pipes P3 - P16 in the table by holding SHIFT to select multiple rows, then click the Always Include in Cost button to move them to the appropriate column to be sized.
3. Now that the pipes have been set to be sized, the Common Size Groups need to be created. Select New above the Pipe Grouping table to create two new groups and give them the names *Headers* and *Distribution*.
4. Add pipes P3 - P6 to the Headers group using the radio buttons. Alternatively, they can be added by selecting the pipes in the Workspace then right-clicking and choosing the appropriate option to add them to the group.
5. Add pipes P7 - P16 to the Distribution group either by using the radio buttons in the table, or from the Workspace.

The Size/Cost Assignments table should now appear as shown in Figure 3.



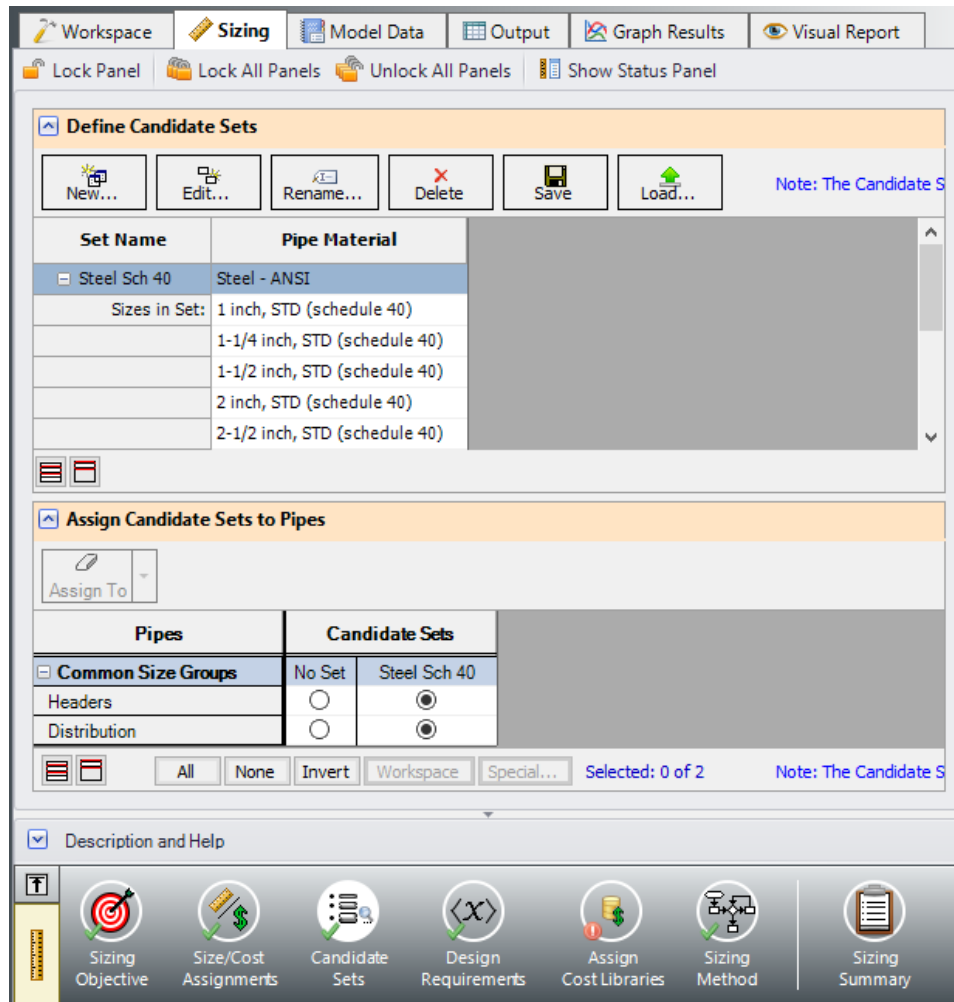
**Figure 3: Size/Cost Assignments window with Common Size Groups defined as outlined above**

### C. Candidate Sets

Click on the Candidate Sets button to open the Candidate Sets window. In this case we will be considering schedule 40 steel pipe within the range of 1" - 12".

1. Select New, and name the Candidate set *Steel Sch 40*.
2. Choose Steel - ANSI from the material list.
3. Expand the STD type, then double click each of the sizes from 1 inch to 12 inch to add them to the Candidate Set.
4. Click OK to accept the defined set.

In the bottom section of the window make sure that the Common Size Groups are assigned to use the new Steel Sch 40 Candidate Set. The window should appear as shown in Figure 4.



**Figure 4: Candidate Sets window fully defined for the model**

## D. Design Requirements

Select the Design Requirements button.

In the primary design case the following requirements need to be met:

1. The flow to the burners must remain below the maximum velocity requirement of 200 feet/sec.
2. All control valves must have a minimum pressure drop of 25 psid.

Make sure that the Pipes button is selected, then create a new Design Requirement named *Max Velocity* specifying the Velocity Outlet Maximum of the pipe is 200 feet/sec. In the bottom section of the window, make sure that the velocity design requirement is applied to each of the discharge lines to the burners (P12 - P16). The Pipe Design Requirements should now appear as shown in Figure 5.

Next select the Control Valves button at the top of the window and create a new control valve Design Requirement. Name the Design Requirement *Min Pressure Drop*, and specify the Pressure Drop Static as a Minimum of 25 psid. In the bottom section apply the Design Requirement to each of the control valves in the model (Figure 6).

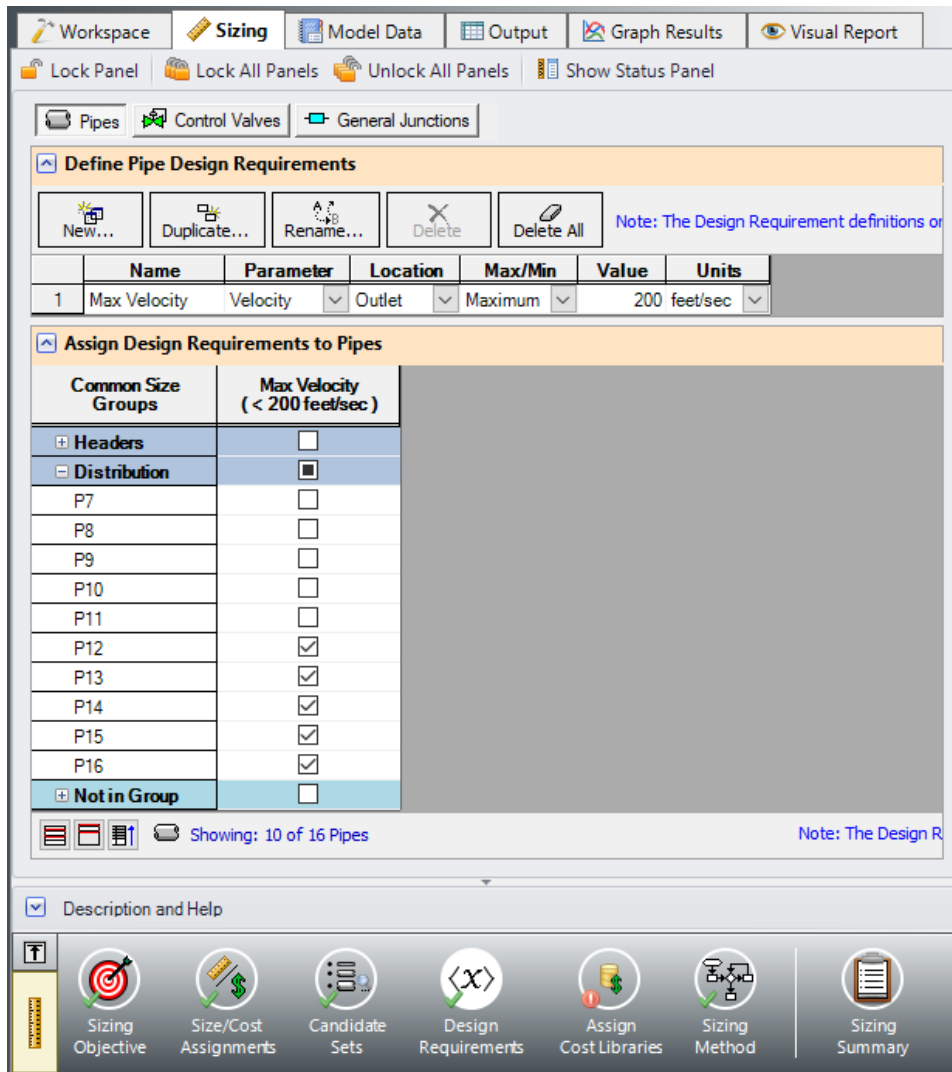


Figure 5: Pipe Design Requirements defined for the primary design case

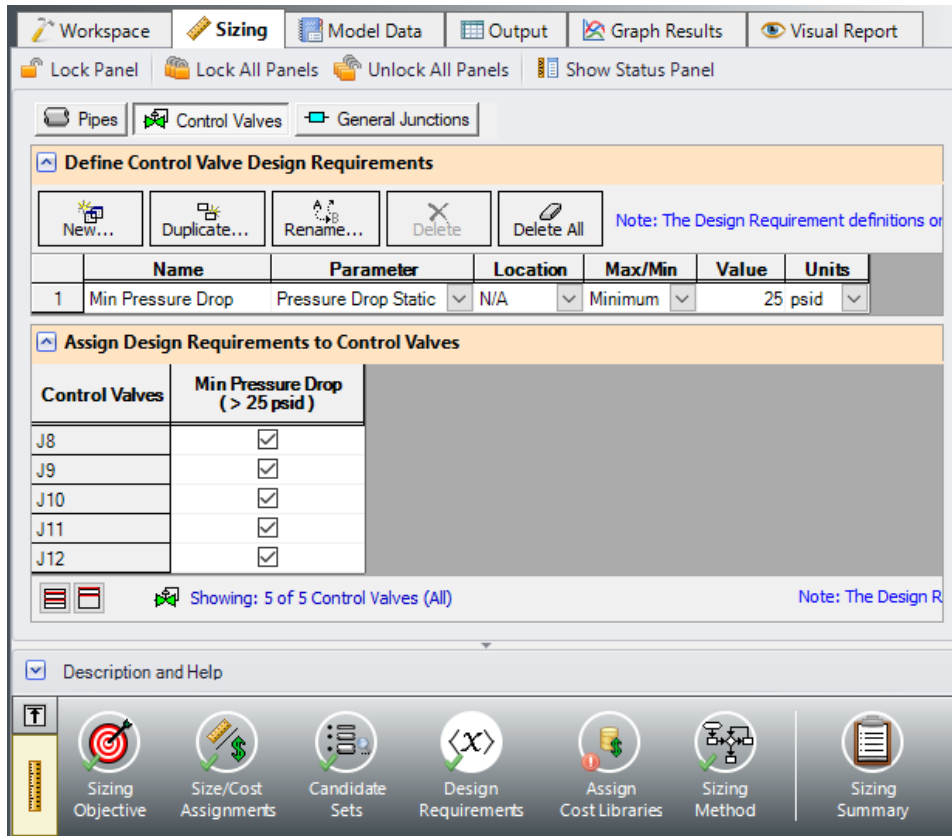


Figure 6: Control Valve Design Requirements defined for the primary design case

## E. Assign Cost Libraries

Select the Assign Cost Libraries button. 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 Steel material costs. To connect the library do the following:

1. Click the button Add Existing Cost Library.
2. Browse to the AFT Arrow 9 Examples folder (located by default in C:\AFT Products\AFT Arrow 9\Examples), and select the file titled **Steel - ANSI Pipe Costs.cst**.

The new library should now be visible in the Available Libraries list, be automatically connected to the model, as shown in Figure 7. The cost library for the pipes should appear with the name Steel - ANSI Pipe Costs and be indented under the Steel - ANSI library within the standard pipe material libraries list.

Once you have confirmed that the library is connected, click Close to exit the Library Manager.

Back in the Assign Cost Libraries panel for the Pipes the Steel - ANSI Pipe Costs library should be the only available cost library, and should be selected to show that it is being applied to the candidate sets and to each of the supply lines as shown in Figure 8.

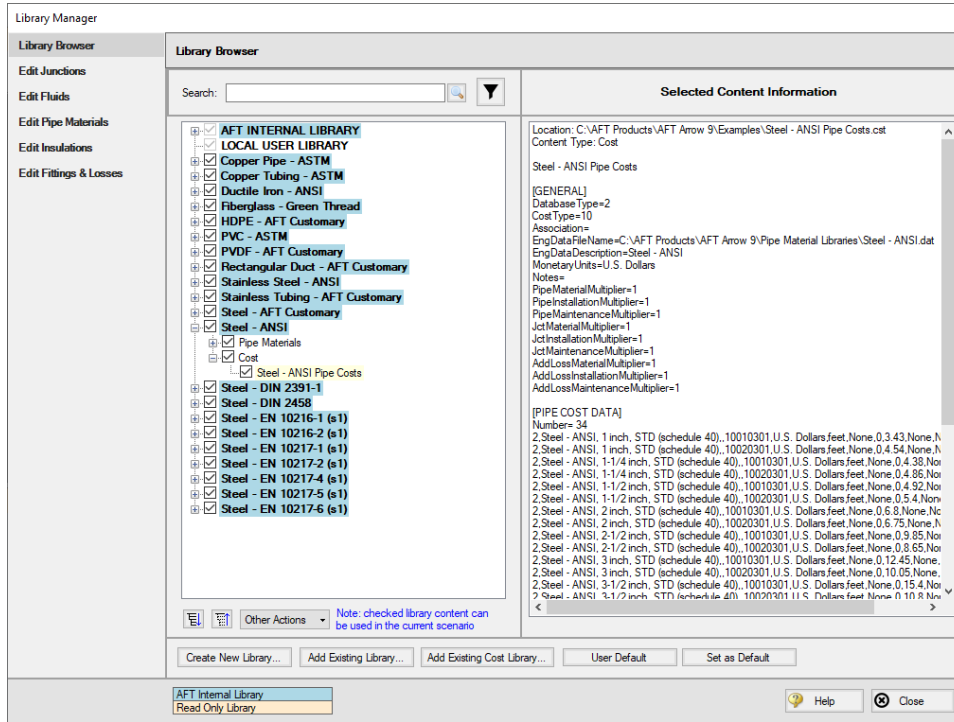
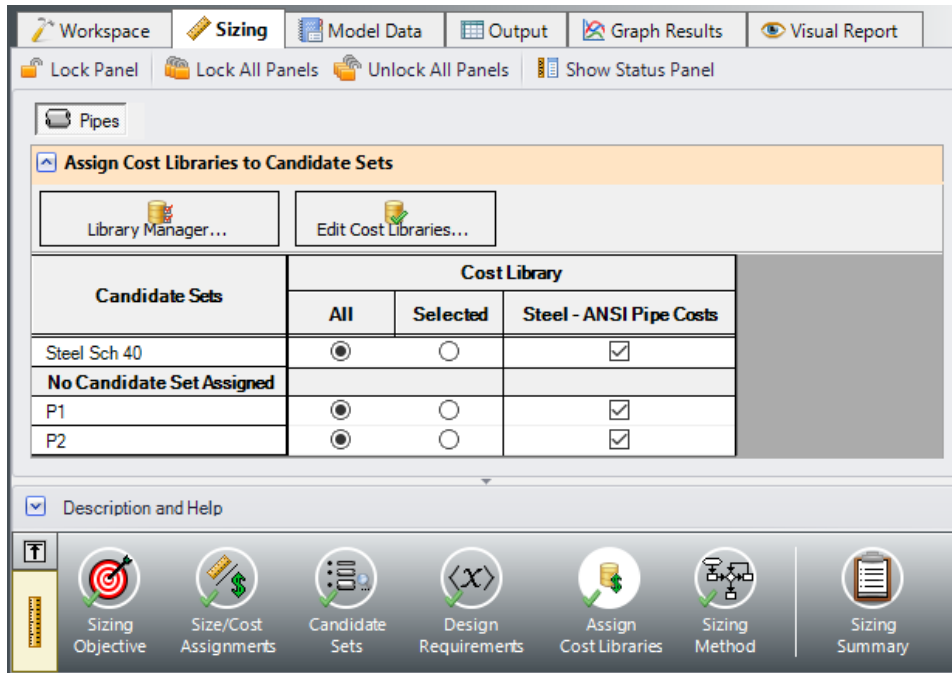


Figure 7: Library Manager with pipe material cost library connected



**Figure 8: Assign Cost Libraries panel with the Steel - ANSI Pipe Costs library connected and applied to the pipes in the model**

## F. Sizing Method

Select the Sizing Method button 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 case where the third burner is shut off. There are a few changes that will need to be made for this additional case:

1. The flow rate at the control valves increases to 6.25 lbm/sec to maintain the overall system flowrate.
2. Control Valve J110 is closed and the minimum design requirement is removed because it is shut off.

To make the dependent design case where the third burner is shut off complete the following:

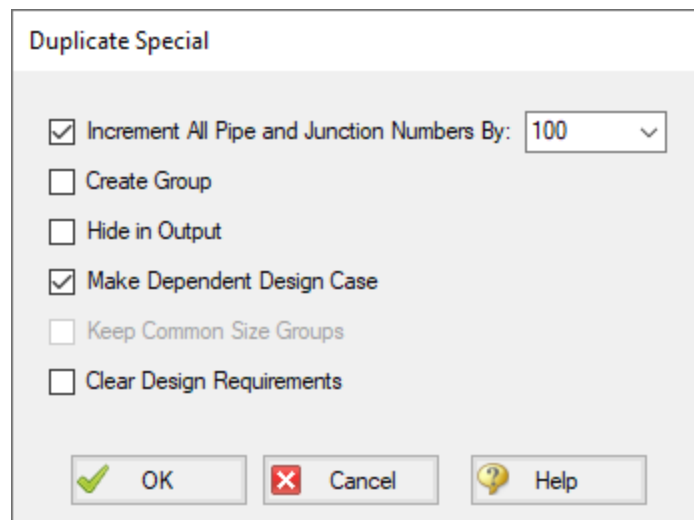
1. Go to the Sizing Objective panel and select the Enable Dependent Design Cases option. A new button will now be available for the Dependent Design Cases panel in the Sizing Navigation panel.

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 case.
3. Go to the Workspace and choose Select All from the Edit menu.
4. Open Duplicate Special (from the Edit menu), enter an increment of 100 and select Make Dependent Design Case (Figure 9). Click OK.
5. Move the duplicated pipes and junctions to distinguish them from the original ones in the Primary Design Case.
6. Close Control Valve J110 in the DDC by selecting the junction in the Workspace and clicking the Special Conditions button  on the Toolbar and selecting Closed.
7. Use Global Junction Edit (from the Edit menu) to change all of the control valve setpoints in the DDC from 5 lbm/sec to 6.25 lbm/sec (see Figure 10).
8. In the Sizing window navigate to the Design Requirements panel. On the Control Valves page remove the pressure drop requirement from J110, since this valve is closed for the DDC.

Return to the Dependent Design Cases panel.

When Duplicate Special was performed with Dependent Design Case selected, each of the duplicated pipes was created with a special type of grouping. For example, pipe 101 is grouped with pipe 1 as a DDC pipe (see Figure 11). 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.

It should also be noted that the dependent design grouping causes each of the dependent pipes to inherit their Common Size Groups, Candidate Sets, and Cost Library settings from the pipes in the primary case. The dependent design case pipes are therefore hidden on all sizing panels except for the Design Requirements and Dependent Design Cases panels.



**Figure 9: Duplicate Special settings to create a dependent design case**

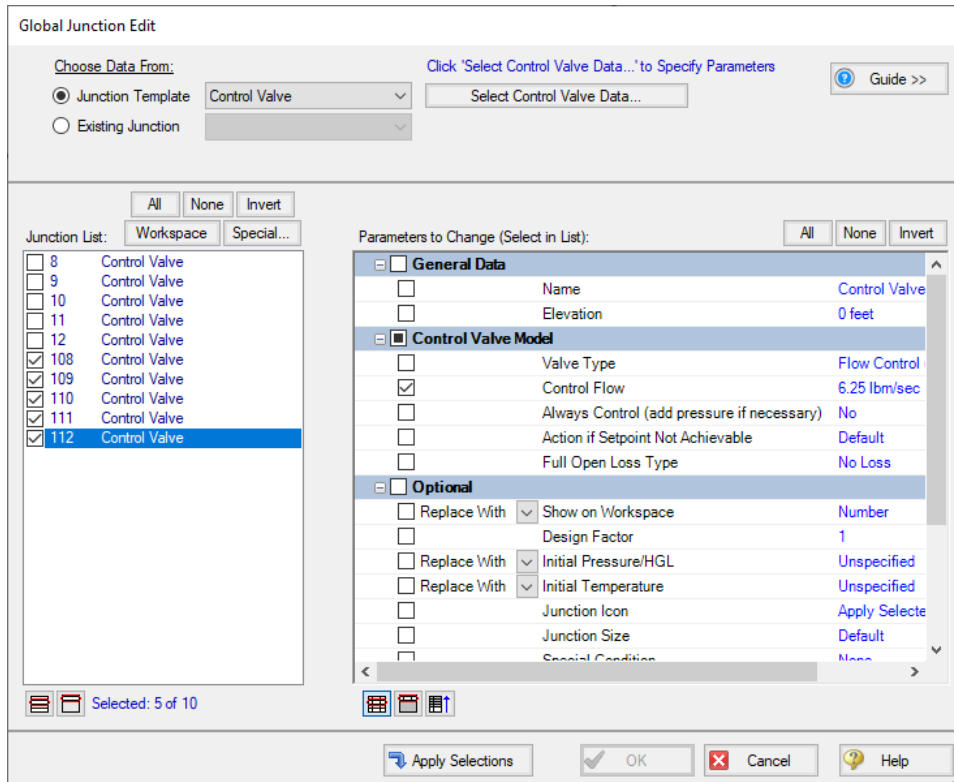
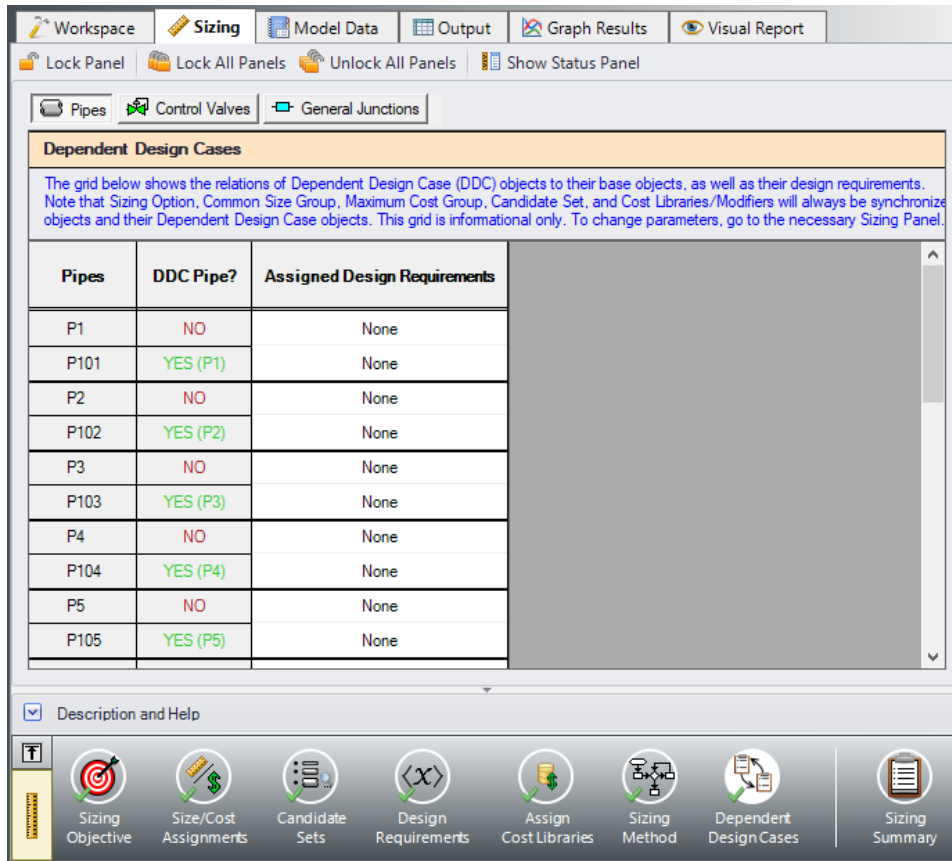


Figure 10: Global Junction Edit is used to change the flows in the homes for the dependent cases





**Figure 11: Pipes in Dependent Design Cases have a special grouping relationship with pipes in the primary case**

## 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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

## Step 5. Examine the Output

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 12 and Figure 13. One can see that the resultant size is 4-inch pipe for the header, with larger 5-inch piping for the distribution lines.

The total cost for the pipes is \$46,946. This includes the \$8,250 cost for the supply line pipes which were not sized, as well as the \$38,696 cost for the pipes in the model which were sized, including the header and distribution piping. The pipes that were sized will be distinguished in the Cost Report by having a green background color.

<span>⤴</span> General Warnings Design Alerts <b>Cost Report</b> Valve Summary <span>⤵</span>				
Table Units: U.S. Dollars		Material	Installation	TOTAL
<b>TOTAL OF ALL MODEL COSTS</b>				<b>46,946</b>
<b>Total of All Shown Costs</b>		<b>28,515</b>	<b>18,431</b>	<b>46,946</b>
<b>Items In Sizing</b>		<b>23,490</b>	<b>15,206</b>	<b>38,696</b>
<b>Items Not In Sizing</b>		<b>5,025</b>	<b>3,225</b>	<b>8,250</b>
<b>Pipe Subtotal</b>		<b>28,515</b>	<b>18,431</b>	<b>46,946</b>
	P1	3,350	2,150	5,500
	P2	1,675	1,075	2,750
o	P3	1,810	1,200	3,010
o	P4	1,810	1,200	3,010
o	P5	1,810	1,200	3,010
o	P6	1,810	1,200	3,010
o	P7	1,950	1,249	3,199
o	P8	1,950	1,249	3,199
o	P9	1,950	1,249	3,199
o	P10	1,950	1,249	3,199
o	P11	1,950	1,249	3,199
o	P12	1,300	833	2,133
o	P13	1,300	833	2,133
o	P14	1,300	833	2,133
o	P15	1,300	833	2,133
o	P16	1,300	833	2,133

*Figure 12: Cost Report for the automated sizing*

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (inches)
1	N/A	N/A	N/A	N/A	N/A
2	N/A	N/A	N/A	N/A	N/A
3	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
4	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
5	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
6	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
7	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
8	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
9	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
10	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
11	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
12	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
13	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
14	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
15	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
16	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047

Figure 13: Sizing Report for the automated pipe sizing

## Step 6. Compare Results With Non-Monetary Sizing

This sizing could be performed by using weight as the objective instead of monetary cost to simplify the setup. Using a non-monetary objective can produce less accurate sizing results in cases where the energy costs or costs of equipment such as compressors need to be accounted for. However, only the pipe sizes are being considered for the sizing calculations in this example. Set up the model to perform a non-monetary sizing by doing the following:

1. In the Scenario Manager create a child of the Base Scenario by right-clicking the scenario name and choosing Create Child.
2. Give the new child scenario the name *Pipe Weight Sizing*.
3. Go to the Sizing Objective panel and change the Objective from Monetary Cost to Pipe Weight.

The model should now be configured to run the model. Click Run Model, then go to the Output window once the run is finished. The final pipe sizes should now appear as shown in Figure 14.

It can be seen that changing the objective to pipe weight produces identical sizing results as the run using a monetary cost objective. For more complex systems this will not always be the case.

Pipe	Name	Sized - Material	Sized - Nominal Size	Sized - Type/Schedule	Sized - Hyd. Diameter (inches)
1	N/A	N/A	N/A	N/A	N/A
2	N/A	N/A	N/A	N/A	N/A
3	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
4	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
5	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
6	Pipe	Steel - ANSI	4 inch	STD (schedule 40)	4.026
7	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
8	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
9	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
10	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
11	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
12	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
13	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
14	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
15	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047
16	Pipe	Steel - ANSI	5 inch	STD (schedule 40)	5.047

Figure 14: Final pipe sizes for the model using a non-monetary cost objective

# Process Steam System - ANS

## Summary

---

This example sizes a process steam system by minimizing the pipe weight. Common Size Groups allow for the system to be simplified while still effectively sizing the system.

---

**Note:** This example can only be run if you have a license for the ANS module.

---

## Topics Covered

- Defining Common Size Groups
- Viewing Common Size Groups using color in the Workspace
- Choosing a Search Method

## Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

In addition, it is assumed that the user has worked through the [Beginner: Three Tank Steam System - 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 Arrow installation:

- *US - Process Steam System.aro*
- *US - Process Steam System - ANS.aro*

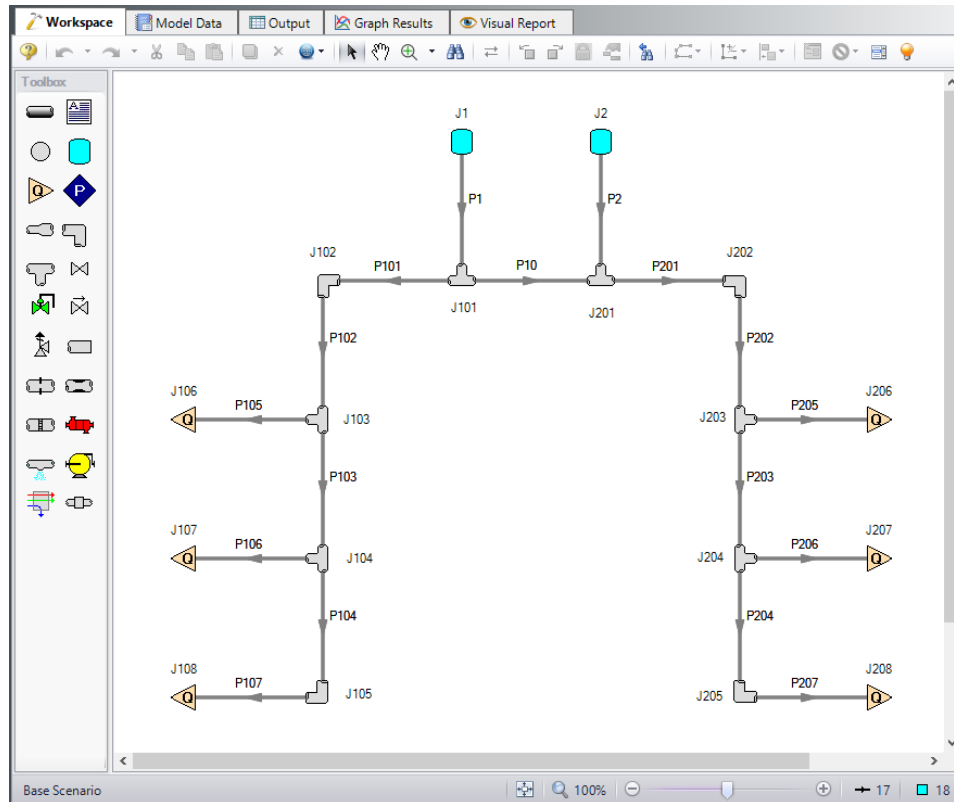
## Step 1. Start AFT Arrow

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow settings, unless you are specifically instructed to do otherwise.

Open the **US - Process Steam System.aro** example file and save it to a new location. Open the newly saved model file, right-click on the Base Scenario and select Delete All Children. The Workspace should look like Figure 1 below.



**Figure 1: Model layout for normal system operation**

## Step 2. Define the Pipes and Junctions Group

Before running this model with ANS, change all Tee/Wye junctions to use the Simple (no loss) Loss Model rather than the Detailed Loss Model. This is significantly reduce the complexity of the model and allow the solver to converge much faster. Use Global Junction Edit to make this change.

## Step 3. Define the Modules Group

Navigate to the Modules panel in Analysis Setup. Check the box next to Activate ANS. The Network option should automatically be selected, making ANS enabled for use.

## Step 4. Define the Automatic Sizing Group

---

The model is already defined for a regular AFT Arrow run, but we need to complete the sizing settings before running the analysis. Go to the Sizing window by clicking the Sizing tab.

### A. Sizing Objective

Go to the Sizing window by selecting the Sizing tab. 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. Select Perform Sizing for this calculation.
2. For the Objective, choose Pipe Weight, and select the Minimize option from the drop-down list.

The Sizing Objective window is now complete.

### B. Sizing Assignments

On the Sizing Navigation panel select the Sizing Assignments button.

For this system, we will be sizing all of the pipes since it is a new system.

➤ **Move all of the pipes** to Always include in Weight.

To more efficiently size the system, it would be beneficial to create common size groups. In this system there are several sections of piping which would likely be grouped together. Create Common Size Groups as described below and indicated by the colors in Figure 2.

1. The boilers in the model are identical, so it would be logical to use equivalent pipe sizes for pipes P1 and P2 connected to the boilers. Create a *Boilers* Common Size Group and add pipes P1 and P2 to the group.
2. Using a common pipe size for the headers would be desirable in this system to reduce installation costs and reduce losses. However, grouping the headers downstream of users 1 and 4 separately could allow us to use a smaller size for those pipes and save costs. Create two groups, *Mains* and *Headers* as shown in yellow and green respectively in Figure 2.
3. In this system the users have varying flow demands, so it is not desirable to use a common pipe size for all of the users. However, several of the users do have identical flows, so the pipes connected to those users should be grouped. Create a Low Flow Users group for pipes P105 and P206 and a High Flow Users group for pipes P107 and P207.

The completed pipe Sizing Assignments can be seen in Figure 3.

# Process Steam System - ANS

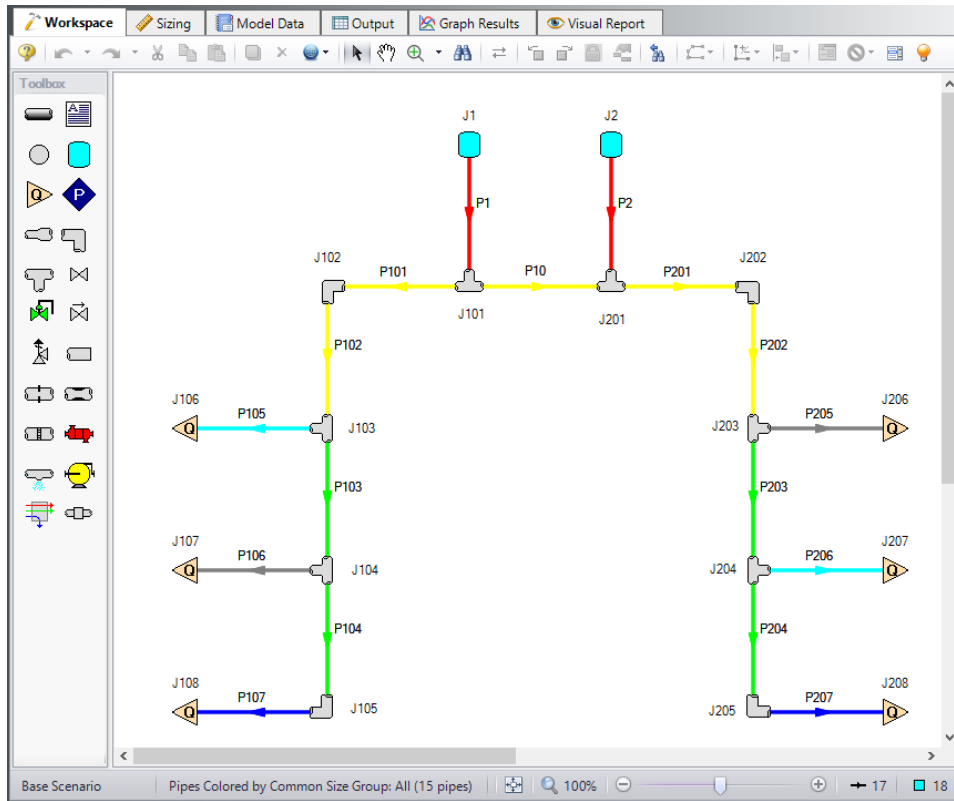


Figure 2: Process Steam model Workspace with the Common Size Groups indicated by color

Pipes	Sizing and Weight Options				Not in Group	Pipe Grouping				
	Automatically Size		Do Not Size			Common Size Groups				
	Always Include in Weight	Include in Weight if Pipe Changes	Include in Weight	Do Not Include in Weight		Boilers	Mains	Headers	Low Flow	High Flow
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"/>
P10	<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"/>
P101	<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"/>
P102	<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"/>
P103	<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"/>
P104	<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"/>
P105	<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"/>
P106	<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"/>
P107	<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"/>
P201	<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"/>
P202	<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"/>
P203	<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"/>
P204	<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"/>
P205	<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"/>
P206	<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"/>
P207	<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"/>

Figure 3: Sizing Assignments fully defined for the model

## C. Candidate Sets



Click on the Candidate Sets button to open the Candidate Sets panel.

For this system it is desired to use STD Steel - ANSI. To create a Candidate Set, do the following:

1. Under Define Candidate Sets, click New.
2. Give the set the name *Schedule 40 Pipes* and click OK.
3. From the drop down list choose Steel - ANSI.
4. In the Available Material Sizes and Types on the left, expand the schedule 40 pipe sizes list.
5. Double-click each of the sizes from 1 to 12 inches to add them to the list on the right.
6. In the Select Pipe Sizes window, click OK.

We now need to define which pipes will use this Candidate Set during the sizing calculation. Check the boxes next to each of the pipes and Common Size Groups to assign the Candidate Set to each of the pipes, as shown in Figure 4.

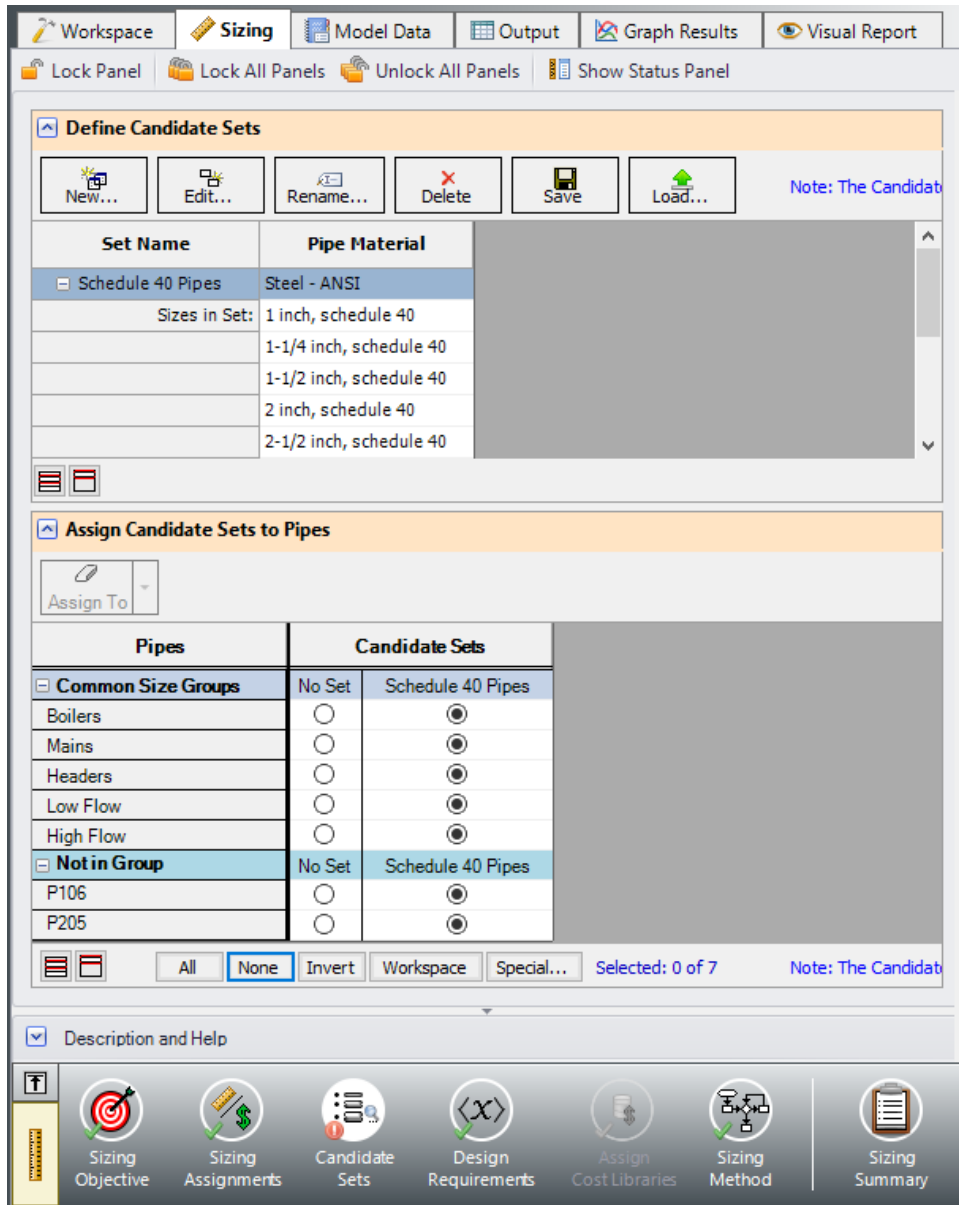


Figure 4: Candidate Sets panel fully defined for the model

## D. Design Requirements

Select the Design Requirements button from the Navigation Panel.

For this system there is a requirement to maintain a minimum discharge pressure of 150 psig to each of the customers. Design Requirements can not be applied on the assigned flow junctions directly, so we will define this requirement at the outlet of each of the pipes connected to the Assigned Flow junctions.

1. Click the New button under Define Pipe Design Requirements.
2. Enter the name *Min Discharge Press*.
3. Select Pressure Static as the Parameter.
4. Set the Location as Outlet.
5. Choose Minimum for Max/Min, and enter 150 psig.

Now we need to apply the defined Design Requirement to the discharge pipes.

➤ **Check the box next to each of the discharge pipes** in the Assign Design Requirements to Pipes section. This can be done quickly by checking the boxes next to the two Common Size Groups and the Not in Group header to select all of the pipes under those classifications. The Design Requirements panel should now appear as shown in Figure 5.

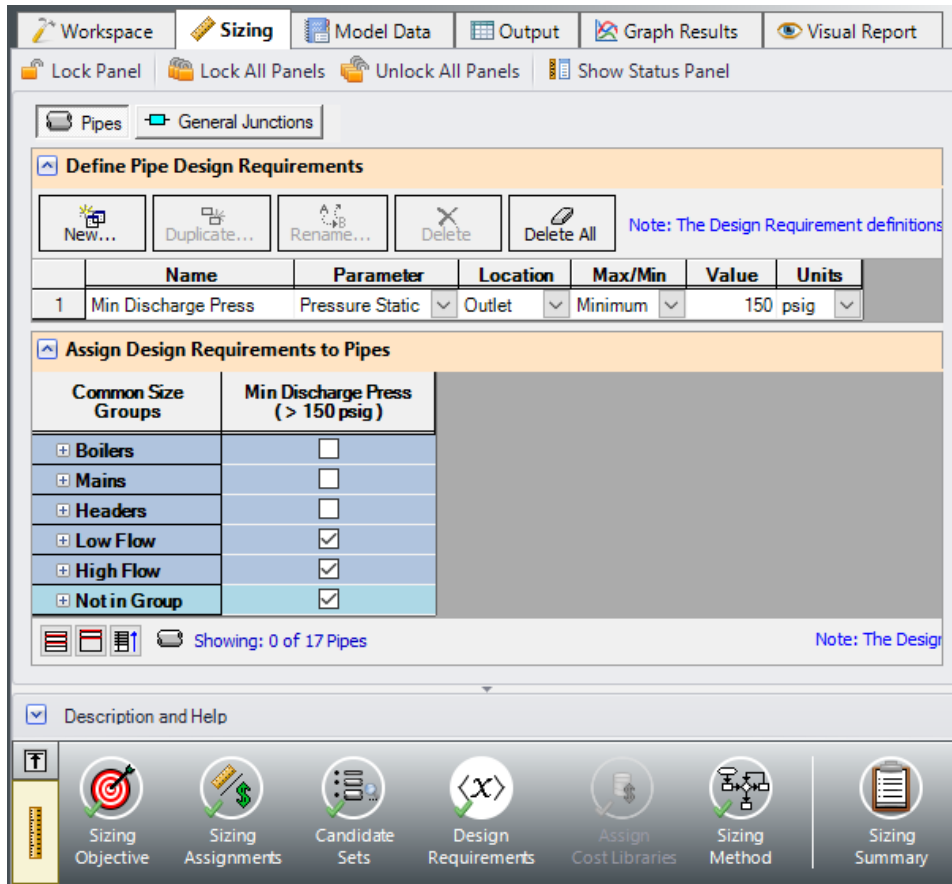


Figure 5: Design Requirements panel for pipes

## E. Assign Cost Library

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 Library panel. Since we have

defined the objective as Pipe Weight, we will not need to assign any cost libraries, and this button is disabled.

## F. Sizing Method

Select the Sizing Method button to go to the Sizing Method panel.

➤ **Choose Discrete Sizing**, since it is desired to select discrete sizes for each of the pipes in the model.

For the search method, the ANS module suggests using the Modified Method of Feasible Directions, so select that option from the Search Methods list. Note that the suggested method will be displayed if the Suggest Best Method button is selected under the Search methods list.

## 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 Arrow solver converges on the answer. Once the solver has converged, view the results by clicking the Output button at the bottom of the Solution Progress window.

## Step 6. Examine the Output

---

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 6. The resultant pipe sizes range from 3-1/2 to 10 inches throughout the system.

With the Modified Method of Feasible Directions the total pipe weight in the system is minimized to 148,621 lbm.

For this system it is possible to further minimize the weight in the system by using the Genetic Algorithm (GA) method. With Genetic Algorithm a further reduction of about 5,000 lbm can be obtained, though the run time is much longer. Note that it is possible to pause the solver and accept the current sizing solution before the Genetic Algorithm has fully completed by clicking Pause, then selecting Accept Sizing from the Other Actions menu and clicking Continue to finish the current iteration. This will provide a valid system design, though it may not be the best solution that could possibly be found.

For more complex models this additional run time may be much longer, but could potentially provide a larger amount of savings. The main product Help file provides more information on setting up the model to use the Genetic Algorithm Search Method.

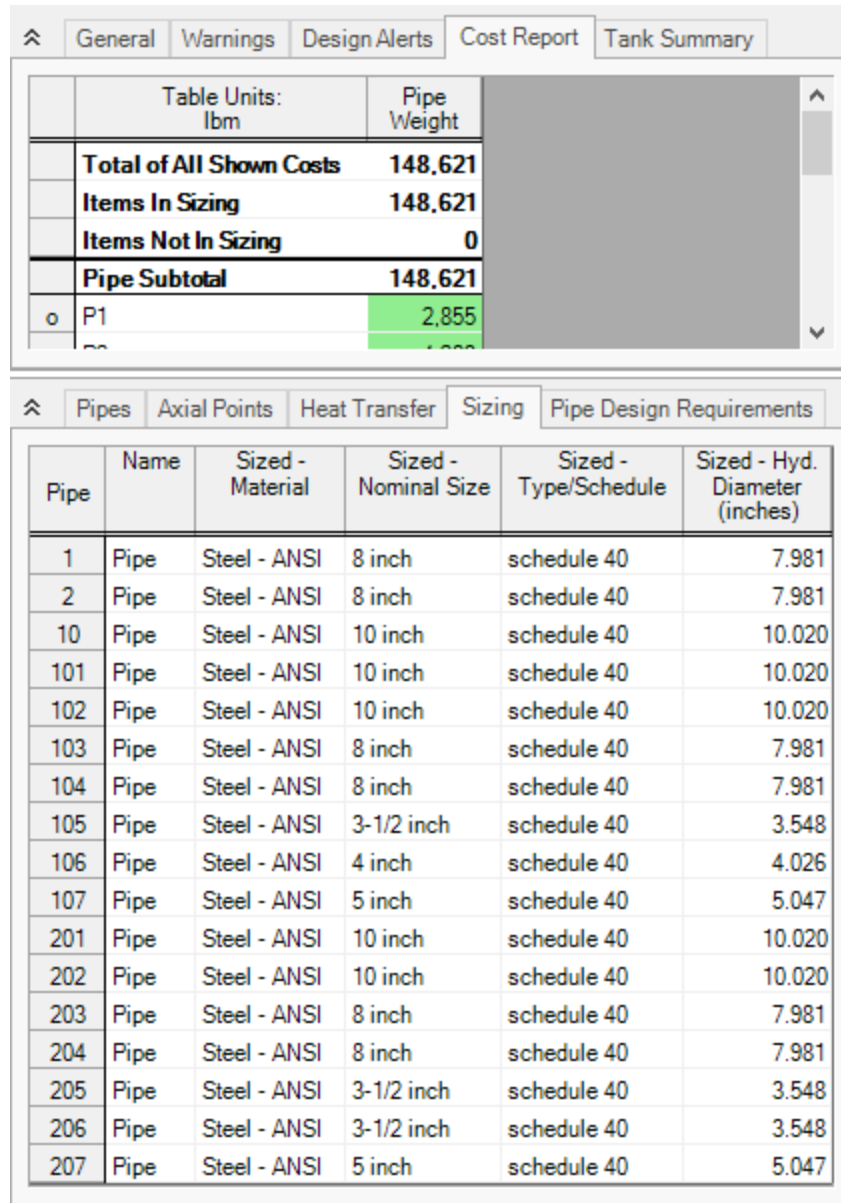


Figure 6: Cost Report and pipe sizes for the sized process steam model

## Extended Time Simulation (XTS) Examples

The examples below utilize the Extended Time Simulation module.

Example	Complexity	Fluid	Pipes	Compressors	Description
<a href="#">Air Receiver Tank</a>	Beginner	Air	4	1	Learn the basics of the XTS module such as defining a transient junction and specifying

Example	Complexity	Fluid	Pipes	Compressors	Description
					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.

## Air Receiver Tank - XTS

### Summary

---

The objective of this example is to introduce the user to the Extended Time Simulation (XTS) module by modeling the pressurizing of an air receiver tank. A positive displacement compressor is used to charge the tank and is shut off once the required pressure is reached for a compressed air process.

---

**Note:** This example can only be run if you have a license for the XTS module.

---

### Topics Covered

- Specifying the Transient Control group
- Understanding Transient Output

### Required Knowledge

This example assumes the user has already worked through the [Beginner: Air Heating System](#) example, or has a level of knowledge consistent with that topic. You can also watch the AFT Arrow [Quick Start Video \(English Units\)](#) on the AFT website, as it covers the majority of the topics discussed in the Beginner: Air Heating System example.

### Model File

This example uses the following file, which is installed in the Examples folder as part of the AFT Arrow installation:

- *US – Air Receiver Tank - XTS.aro*

### Step 1. Start AFT Arrow

---

From the Start Menu choose the AFT Arrow 9 folder and select AFT Arrow 9.

To ensure that your results are the same as those presented in this documentation, this example should be run using all default AFT Arrow 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. Note that XTS can also be enabled from the Analysis menu by going to the Time Simulation option and

selecting Transient, as shown in Figure 1. When XTS is activated a new group will appear in Analysis Setup titled Transient Control.

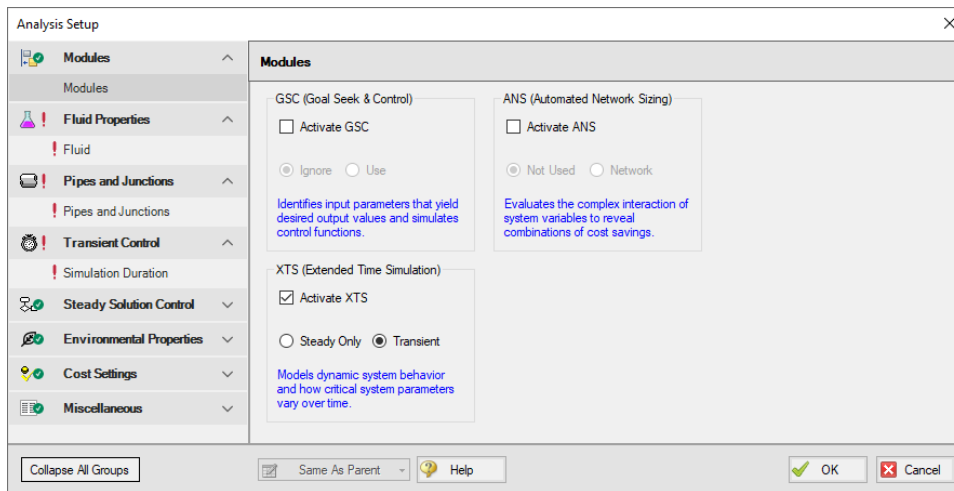


Figure 1: Activating the XTS module and setting it to be enabled in the Modules panel

### 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 = Air
    - i. After selecting, click Add to Model
  - c. Equation of State = Redlich-Kwong
  - d. Enthalpy Model = Generalized
  - e. Specific Heat Ratio Source = Library

### 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.



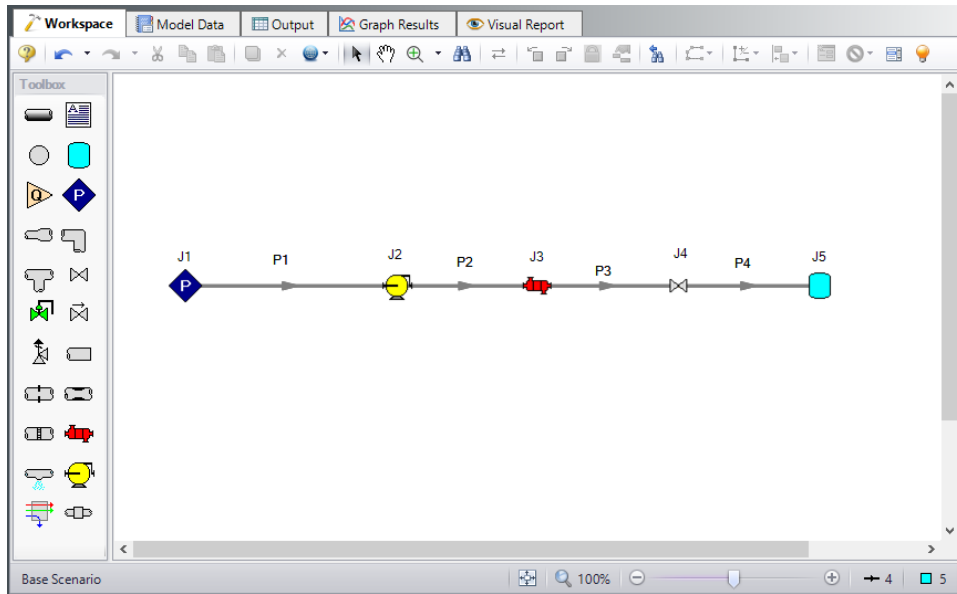


Figure 2: Workspace for XTS Air Receiver Tank example

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.

## Pipe Properties

1. Pipe P1
  - a. Pipe Material = Steel - ANSI
  - b. Pipe Geometry = Cylindrical Pipe
  - c. Size = 3 inch
  - d. Type = STD (schedule 40)
  - e. Friction Model = Standard
  - f. Length = 20 feet
2. Pipes P2 - P4
  - a. Pipe Material = Steel - ANSI
  - b. Pipe Geometry = Cylindrical Pipe
  - c. Size = 3 inch
  - d. Type = STD (schedule 40)
  - e. Friction Model = Standard
  - f. Length = 10 feet

## Junction Properties

1. J1 Assigned Pressure
  - a. Elevation = 0 feet
  - b. Fluid = Air
  - c. Stagnation Pressure = 0 psig
  - d. Stagnation Temperature = 70 deg. F
2. J2 Compressor
  - a. Elevation = 0 feet
  - b. Compressor Model = Positive Displacement Compressor
  - c. Compression Process Thermodynamics = Polytropic
  - d. Polytropic Constant = 1.2
  - e. Compressor Sizing Parameter = Mass Flow Rate
  - f. Fixed Flow Rate = 100 scfm

---

**Note:** The polytropic constant entered for the compressor is intended to represent an active cooling element in the compressor.

---

3. J3 Heat Exchanger
  - a. Elevation = 0 feet
  - b. On the Loss Model Data tab
    - i. Select the Loss Model as Resistance Curve, and then click the Enter Curve Data button
    - ii. Select the Flow Parameter as Mass with units of scfm
    - iii. Enter 100 scfm and 3 psid in the Raw Data table
    - iv. Click Fill as Quadratic to create data for a square loss curve
    - v. Click Generate Curve Fit Now to create a quadratic curve fit
  - c. On the Thermal Data tab:
    - i. Thermal Model = Controlled Downstream Temperature
    - ii. Controlled Outlet Temperature = 85 deg. F
4. J4 Valve
  - a. Elevation = 0 feet
  - b. Loss Model = User Specified
  - c. Subsonic Loss Model = Cv (ANSI/ISA)
  - d. Loss Source = Fixed Cv
  - e. Cv = 50
  - f. Xt = 0.7
5. J5 Tank
  - a. Tank Model = Finite
  - b. Initial Conditions
    - i. Pressure = 0 psig
    - ii. Temperature = 70 deg. F
  - c. Finite Tank Conditions
    - i. Fixed Volume = 500 gal

- ii. Initial Conditions = Known
- iii. Thermodynamic Process = Isothermal

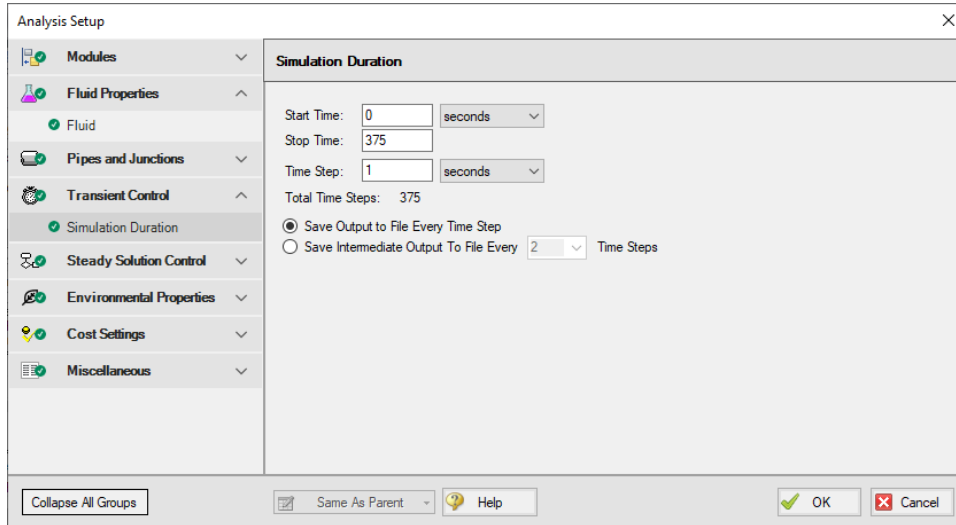
**Figure 3: Tank Properties window defined with Finite Tank option**

➤ **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. Define the Transient Control Group

XTS module transient simulations are controlled through the Simulation Duration panel. You can access the Simulation Duration panel by selecting Transient Control from the Analysis Menu, navigating to the Simulation Duration item in Analysis Setup, or by clicking the Simulation Duration shortcut on the Common Toolbar. When the time simulation mode is set to Steady-Only AFT Arrow will ignore the input in the Simulation Duration panel. When AFT Arrow is set to perform a transient analysis, the Transient Control group becomes a required item before the model can run. Use the Simulation Duration panel to specify transient start and stop times, set time steps, specify the number of output points that are saved to the transient output file (allows XTS to do transient calculations with a small time step while limiting the size of the transient data output file), and adjust solution parameters for finite tank liquid level calculations. Specify start times, stop times, and time steps in any units from seconds to years.

Set the Start Time to 0 seconds and the Stop Time as 375 seconds, with a time step of 1 second. The output will be saved to the output file for every time step. The Simulation Duration panel should appear as shown in Figure 4.



**Figure 4: The Simulation Duration panel is used to control transient simulations**

## Step 6. Set Up the Compressor Transient

In the XTS module, you can define transients for junctions. These transients include events such as valves opening and closing and compressors starting and shutting down. Transients for junctions are defined as either time-based or event-based transients. A time-based transient starts at a specified absolute time. Event based transients initiate when a defined criteria, such as a specified pressure in a particular pipe is reached. There are three types of event transients for junctions: single event, dual event cyclic, and dual event sequential. 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 trigger occur during a simulation (such as a valve closing then reopening). Dual sequential transient events are two transient events which occur one after the other, without repeating.

For this example, set a single event-based transient on the compressor so the compressor will shut off over 30 seconds when the pressure at the tank inlet reaches 125 psig.

Open the Compressor Properties window for compressor J2 and navigate to the Transient tab. Under Initiation of Transient, select Single Event and enter the Transient data as shown below in Figure 5.

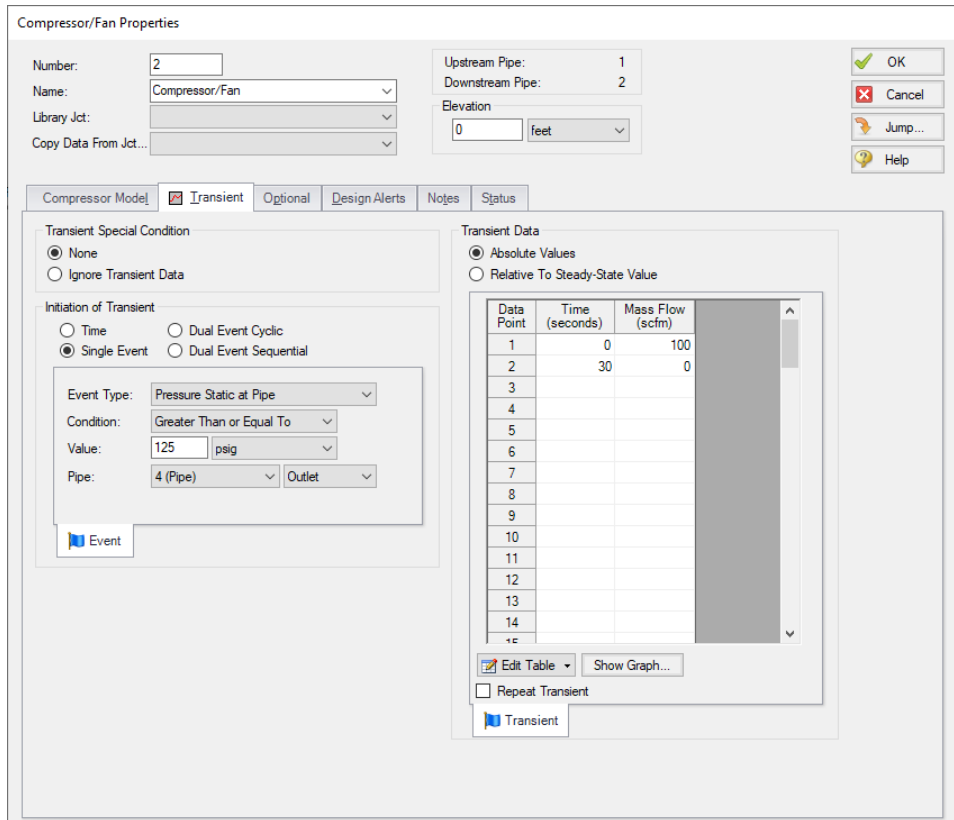


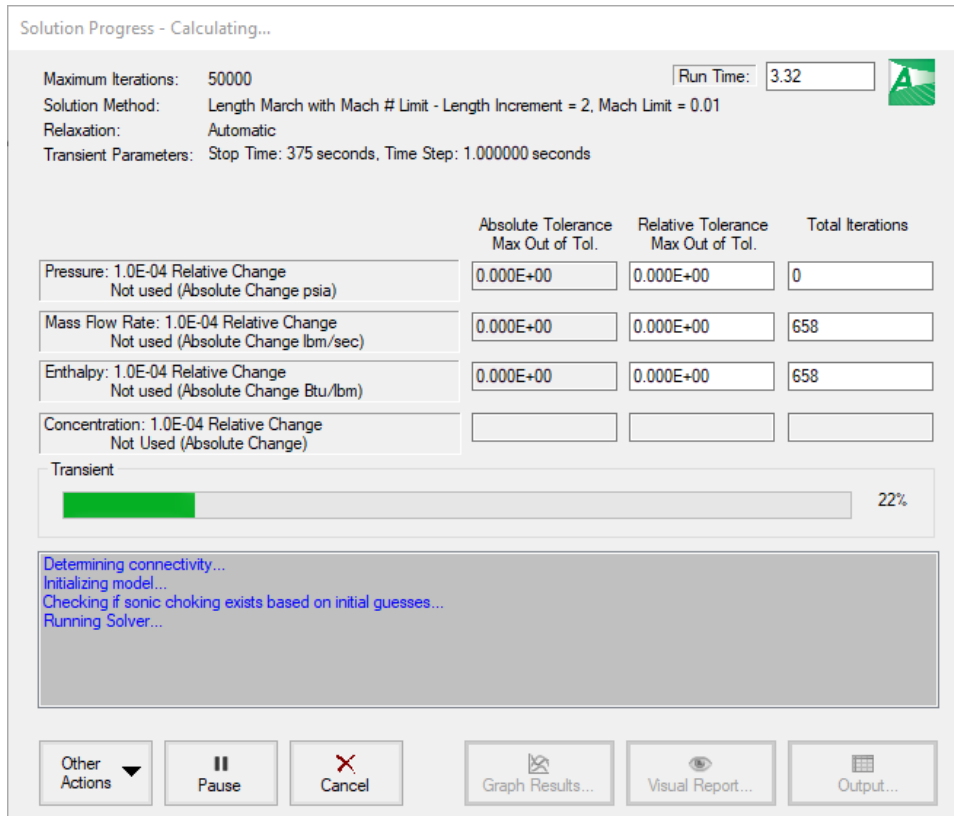
Figure 5: Compressor junction transient tab with flow transient defined

## Step 7. Run the Model

After defining all of the pipes and junctions transient data and transient control items, the transient model can run. Click Run Model from the Common Toolbar or Analysis menu.

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 6.

After the run finishes, view the results by clicking the Output button.

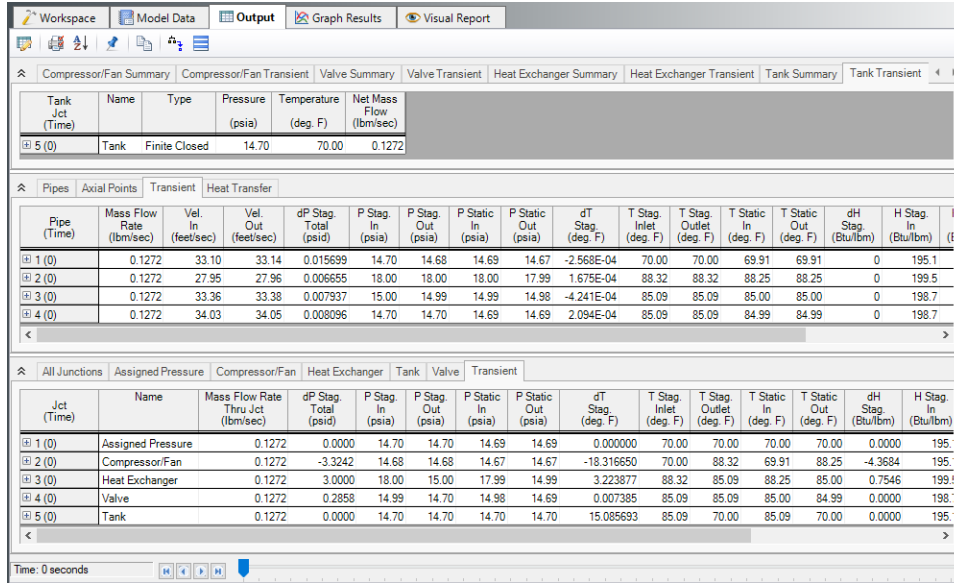


**Figure 6: Solution Progress Window shows the convergence progress, as well as the transient solution progress when running transient cases**

## Step 8. Examine the Transient Results

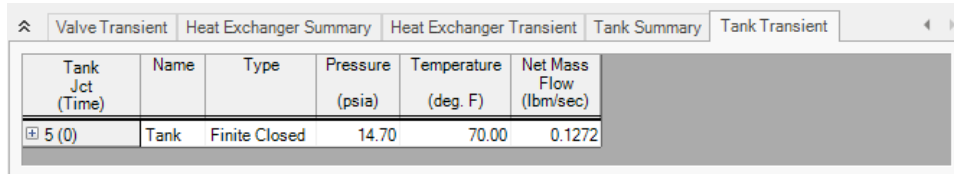
AFT Arrow displays transient output for all Pipes and Junctions in the Output Window, as shown in Figure 7.

# Air Receiver Tank - XTS



**Figure 7: 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 8 shows the Tank Transient summary tab, with all of the data sets collapsed. Figure 9 shows the Tank Transient summary tab, with the data for the J5 Tank expanded.



**Figure 8: Transient Summaries are displayed in the General Output section**

## Air Receiver Tank - XTS

Tank Jct (Time)	Name	Type	Pressure (psia)	Temperature (deg. F)	Net Mass Flow (lbm/sec)
5 (0)	Tank	Finite Closed	14.70	70.00	0.1272
5 (1)	Tank	Finite Closed	15.08	70.00	0.1272
5 (2)	Tank	Finite Closed	15.45	70.00	0.1272
5 (3)	Tank	Finite Closed	15.83	70.00	0.1272
5 (4)	Tank	Finite Closed	16.20	70.00	0.1272
5 (5)	Tank	Finite Closed	16.57	70.00	0.1272
5 (6)	Tank	Finite Closed	16.95	70.00	0.1272
5 (7)	Tank	Finite Closed	17.32	70.00	0.1272
5 (8)	Tank	Finite Closed	17.69	70.00	0.1272

**Figure 9: Transient Summary data can be expanded to show the junction data for every Time Step**

The Event Messages tabs in the General Section (Figure 10) can be used to see when events are triggered during the transient. For this example the Event Messages By Jct tab shows that the flow transient at the compressor was triggered at 335 seconds into the transient, meaning it took 335 seconds to pressurize the tank to the required pressure.

\*\*\* Events Sorted By Jct Number \*\*\*

Compressor/Fan Junction 2 first event started at Time: 335 seconds

**Figure 10: Event Messages tab in the General section of the Output window**

The Quick Graph feature can be used to plot transient data for rapid 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 11 illustrates how to generate a Quick Graph for J5 Tank Pressure vs. Time. The graph illustrates how the pressure in the tank rises over time and steadies out at about 145 psia once the compressor fully shuts off (Figure 12). The Graph Results window may also be used to create a full range of graphs related to the XTS module transient output.

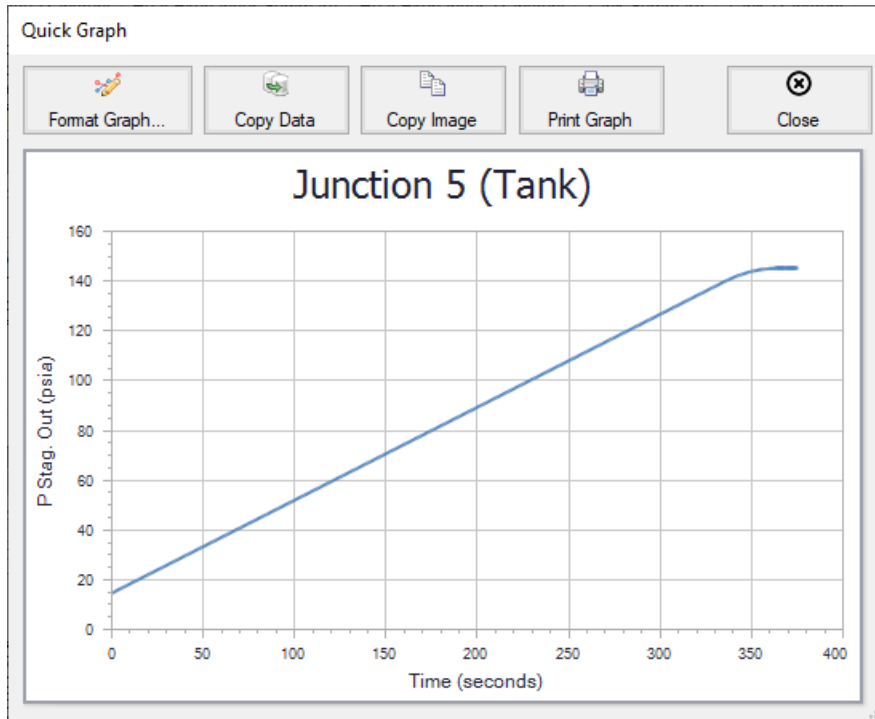
Jct (Time)	Name	Mass Flow Rate Thru Jct (lbm/sec)	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T (deg. F)
1 (0)	Assigned Pressure	0.1272	0.0000	14.70	14.70	14.69	14.69	0.000000	70.00	70.00	
2 (0)	Compressor/Fan	0.1272	-3.3242	14.68	14.68	14.67	14.67	-18.316650	70.00	88.32	
3 (0)	Heat Exchanger	0.1272	3.0000	18.00	15.00	17.99	14.99	3.223877	88.32	85.09	
4 (0)	Valve	0.1272	0.2858	14.99	14.70	14.98	14.69	0.007385	85.09	85.09	
5 (0)	Tank	0.1272	0.0000	14.70	14.70	14.70	14.70	15.085693	85.09	70.00	

Time: 0 seconds

Base Scenario

**Figure 11: Use Quick Graph to easily plot J5 Tank pressure**





**Figure 12: Quick Graph Plot of J5 Tank pressure**

Similar to the summary transient data, the transient tabs in the Pipe and Junctions sections of the Output Window (shown in Figure 7) are used to display the transient data for each pipe and junction at every time step. This transient data can be expanded or collapsed in the same manner as in the General Output transient summaries.

Figure 13 shows the output data for the pipes and junctions, found on the Pipes and All Junctions tabs, at the initial time step, Time = 0 seconds. 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 14 shows the pipe and junction data at Time = 60 seconds.

# Air Receiver Tank - XTS

General

Pipes Axial Points Transient Heat Transfer

Time: 0 Pipe	Name	Mass Flow Rate (lbm/sec)	Vel. In (feet/sec)	Vel. Out (feet/sec)	Mach # In	Mach # Out	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. Inlet (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)
1	Pipe	0.1272	33.10	33.14	0.02933	0.02936	0.015699	14.70	14.68	14.69	14.67	-2.568E-04	70.00	70.00	69.91	69.91
2	Pipe	0.1272	27.95	27.96	0.02435	0.02436	0.006685	18.00	18.00	18.00	17.99	1.675E-04	88.32	88.32	88.25	88.25
3	Pipe	0.1272	33.36	33.38	0.02915	0.02916	0.007937	15.00	14.99	14.99	14.98	-4.241E-04	85.09	85.09	85.00	85.00
4	Pipe	0.1272	34.03	34.05	0.02973	0.02975	0.008096	14.70	14.70	14.69	14.69	2.094E-04	85.09	85.09	84.99	84.99

All Junctions Assigned Pressure Compressor/Fan Heat Exchanger Tank Valve Transient

Time: 0 Jct	Name	Mass Flow Rate Thru Jct (lbm/sec)	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. Inlet (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)	dH Stag. Inlet (Btu/lbm)	H Stag. Inlet (Btu/lbm)	H Stag. Outlet (Btu/lb)
1	Assigned Pressure	0.1272	0.0000	14.70	14.70	14.69	14.69	0.000000	70.00	70.00	70.00	70.00	0.0000	195.1	195.1
2	Compressor/Fan	0.1272	-3.3242	14.68	14.68	14.67	14.67	-18.316650	70.00	88.32	69.91	88.25	-4.3684	195.1	195.1
3	Heat Exchanger	0.1272	3.0000	18.00	15.00	17.99	14.99	3.223877	88.32	85.09	88.25	85.00	0.7546	199.5	199.5
4	Valve	0.1272	0.2858	14.99	14.70	14.98	14.69	0.007385	85.09	85.09	85.00	84.99	0.0000	198.7	198.7
5	Tank	0.1272	0.0000	14.70	14.70	14.70	14.70	15.085693	85.09	70.00	85.09	70.00	0.0000	195.1	195.1

Time: 0 seconds

Base Scenario Cautions Exist 100%

**Figure 13: Pipe and Junction transient data at Time = 0 minutes**

Pipes Axial Points Transient Heat Transfer

Time: 60 Pipe	Name	Mass Flow Rate (lbm/sec)	Vel. In (feet/sec)	Vel. Out (feet/sec)	Mach # In	Mach # Out	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. Inlet (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)
1	Pipe	0.1272	33.10	33.14	0.02933	0.02936	0.015699	14.70	14.68	14.69	14.67	-2.568E-04	70.00	70.00	69.91	69.91
2	Pipe	0.1272	14.30	14.30	0.01166	0.01166	0.003460	40.24	40.23	40.23	40.23	6.417E-05	166.89	166.89	166.87	166.87
3	Pipe	0.1272	13.43	13.43	0.01174	0.01174	0.003193	37.23	37.23	37.23	37.23	9.778E-05	85.02	85.01	85.00	85.00
4	Pipe	0.1272	13.47	13.47	0.01177	0.01177	0.003204	37.12	37.12	37.12	37.11	8.238E-05	85.01	85.01	85.00	85.00

All Junctions Assigned Pressure Compressor/Fan Heat Exchanger Tank Valve Transient

Time: 60 Jct	Name	Mass Flow Rate Thru Jct (lbm/sec)	dP Stag. Total (psid)	P Stag. In (psia)	P Stag. Out (psia)	P Static In (psia)	P Static Out (psia)	dT Stag. Inlet (deg. F)	T Stag. Inlet (deg. F)	T Stag. Outlet (deg. F)	T Static In (deg. F)	T Static Out (deg. F)	dH Stag. Inlet (Btu/lbm)	H Stag. Inlet (Btu/lbm)	H Stag. Outlet (Btu/lb)
1	Assigned Pressure	0.1272	0.0000	14.70	14.70	14.69	14.69	0.000000	70.00	70.00	70.00	70.00	0.00	195.1	195.1
2	Compressor/Fan	0.1272	-25.5567	14.68	14.68	14.67	14.68	-96.886963	70.00	166.89	69.91	166.87	-23.17	195.1	195.1
3	Heat Exchanger	0.1272	3.0000	40.23	37.23	40.23	37.23	81.872009	166.89	85.02	166.87	85.00	19.71	218.3	218.3
4	Valve	0.1272	0.1116	37.23	37.12	37.23	37.12	0.002869	85.01	85.01	85.00	85.00	0.00	198.6	198.6
5	Tank	0.1272	0.0000	37.12	37.12	37.12	37.12	15.012024	85.01	70.00	85.01	70.00	0.00	195.1	195.1

Time: 60 seconds

Base Scenario Cautions Exist 100%

**Figure 14: Pipe and Junction transient data at Time = 60 seconds**

**Index**

---

**A**

Arrow Modules Overview 123

**C**

Compressed Air System - CST 98

**N**

Natrural Gas Burner - GSC 138