AFT Arrow™ 9
Quick Start Guide
Metric Units

Dynamic solutions for a fluid world ™

AFT
Applied Flow Technology

AFT Arrow™
Quick Start Guide

Metric Units

AFT Arrow Version 9

Compressible Pipe Flow Modeling

Published: April 28, 2023

Dynamic solutions for a fluid world ™

CAUTION!

AFT Arrow is a sophisticated pipe flow analysis program designed for qualified engineers with experience in pipe flow analysis and should not be used by untrained individuals. AFT Arrow is intended solely as an aide for pipe flow analysis engineers and not as a replacement for other design and analysis methods, including hand calculations and sound engineering judgment. All data generated by AFT Arrow should be independently verified with other engineering methods.

AFT Arrow is designed to be used only by persons who possess a level of knowledge consistent with that obtained in an undergraduate engineering course in the analysis of pipe system fluid mechanics, have been trained in the specialized field of gas dynamics, and are familiar with standard industry practice in pipe flow analysis.

AFT Arrow is intended to be used only within the boundaries of its engineering assumptions. The user should consult the AFT Arrow Help System for a discussion of all engineering assumptions made by AFT Arrow.

Contents

Introducing AFT Arrow

Welcome to AFT Arrow™ 9, Applied Flow Technology's powerful compressible pipe flow modeling tool. With AFT Arrow you can model compressible flow in complex pipe networks, and include the complicating effects of real gas behavior, heat transfer, gas mixing, and sonic choking at one or more locations. In addition, it includes steam property calculation from the most respected source, the ASME Steam Tables. These capabilities will allow you to properly understand the behavior of your gas system.

AFT Arrow performs a complete mass, momentum and energy balance on your pipe system. It implements a sophisticated solution engine that marches down pipes, offering superior accuracy to commonly used lumped methods. The engineer accesses these capabilities through an advanced graphical interface.

AFT Arrow 9 has three available add-on modules which extend AFT Arrow's extensive modeling capabilities into new areas.

- The Goal Seek & Control (GSC) module allows the engineer to perform multi-variable goal seeking and simulate control system functions.

- The Automated Network Sizing (ANS) module automates the process of sizing pipes and ducts in a complex network based on required conditions, such as maximum pressure or minimum flow, to reduce cost or weight.

- The Extended Time Simulation (XTS) module allows the engineer to model time varying system behavior.

AFT Arrow is a proven tool which provides the engineer with an extremely broad and powerful feature set ready to tackle the most demanding analysis problems.

Modeling capabilities

AFT Arrow can be used to model a wide variety of compressible flow systems. AFT Arrow consists of the standard version plus three optional add-on modules.

AFT Arrow 9 capabilities

- Open and closed (recirculating) systems
- Network systems that branch or loop, with no limit on the number of loops
- Pressure-driven systems
- Compressor or fan-driven systems, including multiple compressor/fans in parallel or in series
- Compressor/fans with variable speed, controlled pressure, controlled flow, and a fixed compression ratio
- Systems with pressure and/or flow control valves
- Systems with valves closed and compressor/fans turned off
- Heat transfer analysis and system energy balance
- Systems with non-ideal gases
- Systems that experience sonic choking, including multiple sonic choking points
- Systems with non-reacting flow stream mixing and user-defined mixtures with NIST REFPROP (or with optional Chempak)
- Systems with elevation changes or rotation such as in turbomachinery

Add-on module capabilities

- Multi-variable goal seeking and control simulation (GSC)

- Automated design of piping networks (ANS)

- Transient modeling (XTS)

- Valves opening and closing over time (XTS)

- Compressors/fans starting and stopping over time (XTS)

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

AFT Arrow provides two independent solution methods that can be run individually or in combination with each other to allow flexibility and cross-verification of results. In addition, AFT Arrow offers two lumped methods that offer improved computation speed with a slight loss in accuracy. Modeling of low to high pressure and low to high velocity systems can be performed. Sonic capabilities allow modeling of high velocity conditions such as in relief systems.

AFT Arrow can model a large range of gas systems including steam, natural gas, air, and chemical and petrochemical process gases. Support for real gases is offered through several generalized equation-of-state models, ASME Steam tables, NIST REFPROP, or the optional Chempak add-on.

Thermophysical property data

AFT Arrow derives physical properties from one of four sources. The first is the standard AFT Arrow set of gases which contains data for 28 common gases (called AFT Standard). The AFT Standard set is user customizable, but does not support gas mixing.

The second option is the ASME Steam library. ASME Steam offers improved accuracy on steam calculations with a corresponding increase in model run time.

The third option is the NIST REFPROP library, first introduced in AFT Arrow 7. NIST REFPROP is licensed from the National Institute of

Standards and Technology and is included in AFT Arrow. NIST REFPROP has a library of approximately 150 fluids.

The fourth option is the Chempak™ library. Chempak is licensed from Madison Technical Software and is offered as an optional add-on to AFT Arrow 9. Chempak has a library of approximately 600 gases.

NIST REFPROP and Chempak support gas mixing in two forms. First, users can create their own mixtures. These are called static mixtures and are defined before the model is run. Second, as flow streams of different composition mix, an entirely new gas mixture results. Its composition depends on the composition of the incoming streams as well as the relative flow rates. AFT Arrow updates the composition of this new mixture as the simulation progresses. For this reason, these mixtures are referred to as dynamic mixtures. AFT Arrow is restricted to non-reacting gas mixture calculations.

Engineering assumptions in AFT Arrow

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

- Compressible flow

- All gases are superheated

- Steady-state conditions

- One-dimensional flow

- No chemical reactions

- Supersonic flow does not exist in the system

AFT Arrow allows you to change the gravitational acceleration of the system, a feature useful for aerospace vehicle applications or pipe systems that are designed for extraterrestrial environments.

AFT Arrow Primary Windows

The AFT Arrow window has five subordinate windows that work in an integrated fashion. Each is located on a separate, moveable tab. You

work exclusively from one of these windows at all times. For this reason, they are referred to as Primary Windows.

Of the five Primary Windows, two are input windows, two are output windows, and one displays output and input information. Figure 1.1 shows the relationship between the Primary Windows.
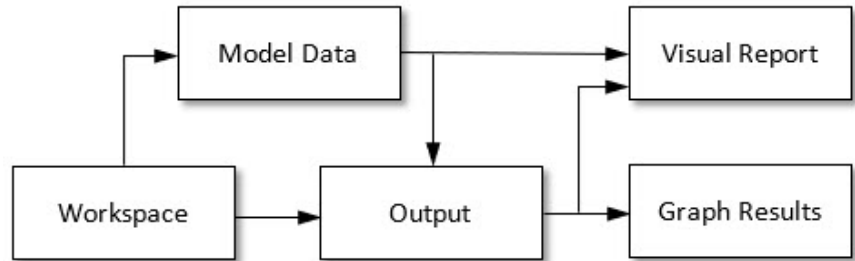
Figure 1.1   Primary Window workflow in AFT Arrow

Input windows

The two windows that function exclusively as input windows are the Workspace window and the Model Data window. These two windows, one graphical and the other text-based, work together to process model input data with immense flexibility. The tools that are provided in these two windows allow you to model a large variety of pipe networks.

The Visual Report window can function in support of both input and output data. As an input window, it allows you to see the input data superimposed on the pipe system schematic created on the Workspace.

Output windows

The two windows that function exclusively as output windows are the Output window and the Graph Results window. The Output window is text-based, while the Graph Results window is graphical. These two windows offer a powerful and diverse range of features for reviewing analysis results for modeling errors, gaining a deeper understanding of the pipe system's flow behavior, and preparing the results for documentation.

As an output window, Visual Report allows you to see the output results superimposed on the pipe system schematic created on the Workspace.

The five Primary Windows form a tightly integrated, highly efficient system for entering, processing, analyzing, and documenting compressible flow analyses of pipe networks.

Note: AFT Arrow will support dual monitors. 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.

Flow Through an Orifice Example

This chapter is intended to give you the big picture of AFT Arrow's layout and structure. Other example model discussions on the AFT website. To get there, open the Help menu and choose Show Examples or go to www.AFT.com.

This example demonstrates a sample calculation to determine the maximum flow through a system where sonic choking occurs.

Topics covered

This example will cover the following topics:

- Model building basics
- Entering pipe and junction data
- Specifying Analysis Setup
- Specifying Output Control
- Creating Visual Reports
- Graphing results

Required knowledge

No prior knowledge is required for this example.

Model file

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

- *Metric - Orifice Flow.aro* - AFT Arrow model file

This example is also provided in English units in the "*US - Orifice Flow.aro*" model file.

This example will require you to build the model from scratch to help familiarize yourself with the steps required to build a complete model in AFT Arrow. Therefore, use this example model file as a reference only.

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 3 meters long and the second pipe leading from the orifice to the discharge tank is 6 meters long. Both pipes are Steel - ANSI, 2 inch, STD (schedule 40), and they are very well insulated (adiabatic).

The inlet tank has a pressure of 16 barG (1600 kPa(g)), and a temperature of 260 deg. C. The discharge tank has a pressure of 0 barG. The elevation of both tanks is 0 meters.

The orifice at the end of the first pipe has an area of 20 cm^2 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 exit?

3.   At the maximum flow, what is the sonic area at the orifice? At the discharge tank?

Step 1. Start AFT Arrow

➢   **To start AFT Arrow**, click Start on the Windows taskbar, and choose AFT Arrow from the AFT Arrow 9 folder. (This refers to the standard

menu items created by a setup. You may have chosen to specify a different menu item).

As AFT Arrow is started, the Arrow Startup window appears, as shown in Figure 2.1. This window provides you with several options you can choose before you start building a model.
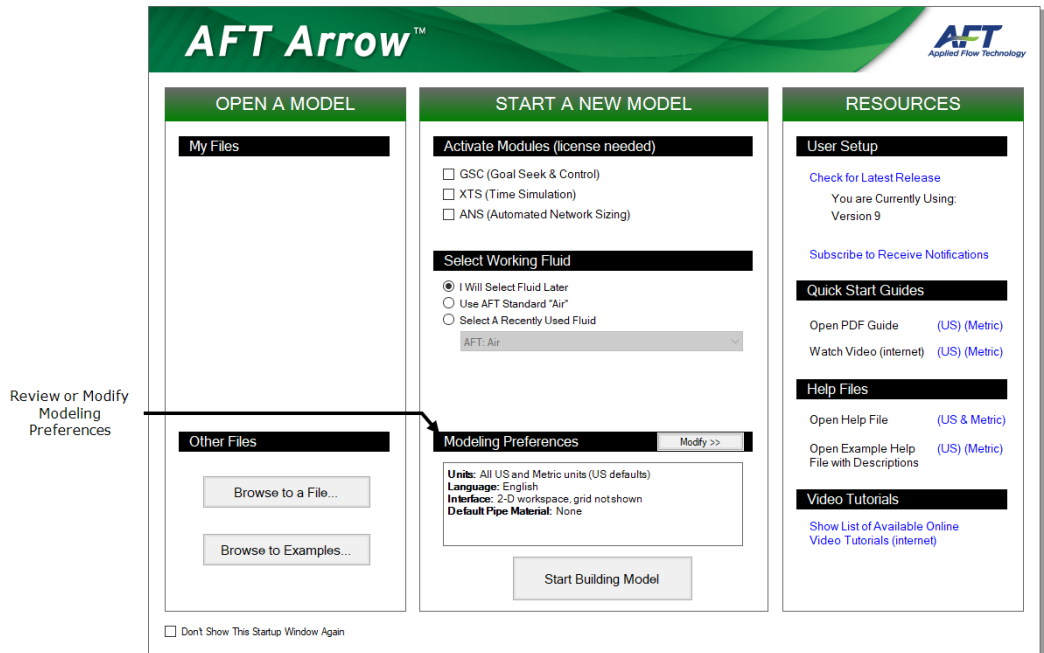
Figure 2.1   Startup Window with Modeling Preferences Collapsed

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

- 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

If this is the first time that you have started AFT Arrow, Modeling Preferences will be expanded in the middle section of the Startup Window, as shown in Figure 2.2. If this is not the first time that you have started AFT Arrow, the Startup Window will appear with Modeling Preferences collapsed, as shown in Figure 2.1.

When collapsed, you can view your current Modeling Preferences at the bottom of Start a New Model. To further review or adjust your preferences, click the "Modify>>" button (see Figure 2.1).
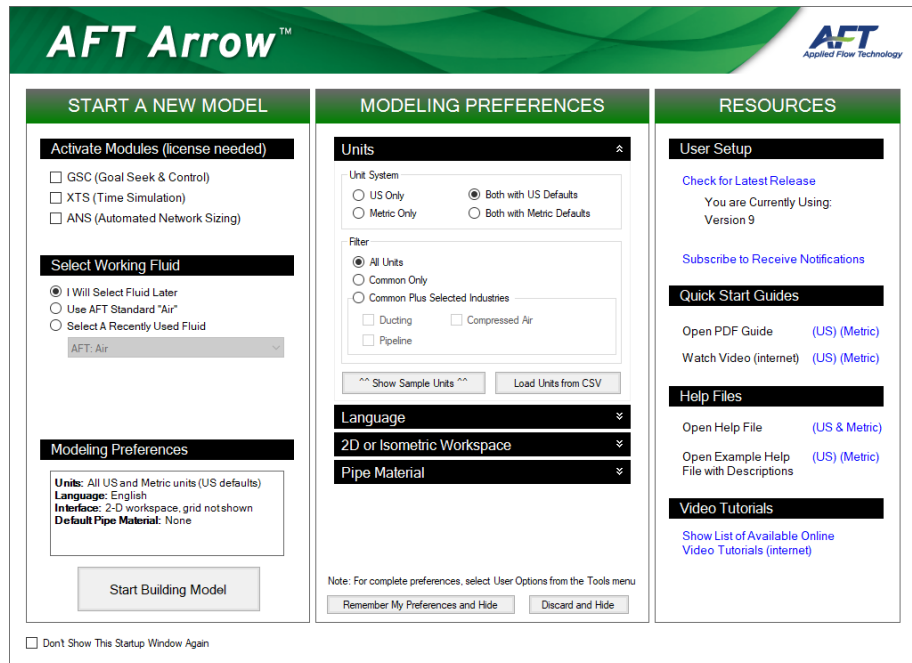
Figure 2.2   Startup Window with Modeling Preferences Expanded

With Modeling Preferences expanded, as in Figure 2.2, select Both with Metric Defaults under Unit System. Select "Common Only" under Filter to show only commonly used units, instead of "All Units."

You can "Show Sample Units" to see which units will be included based on your selections, as shown in Figure 2.3.
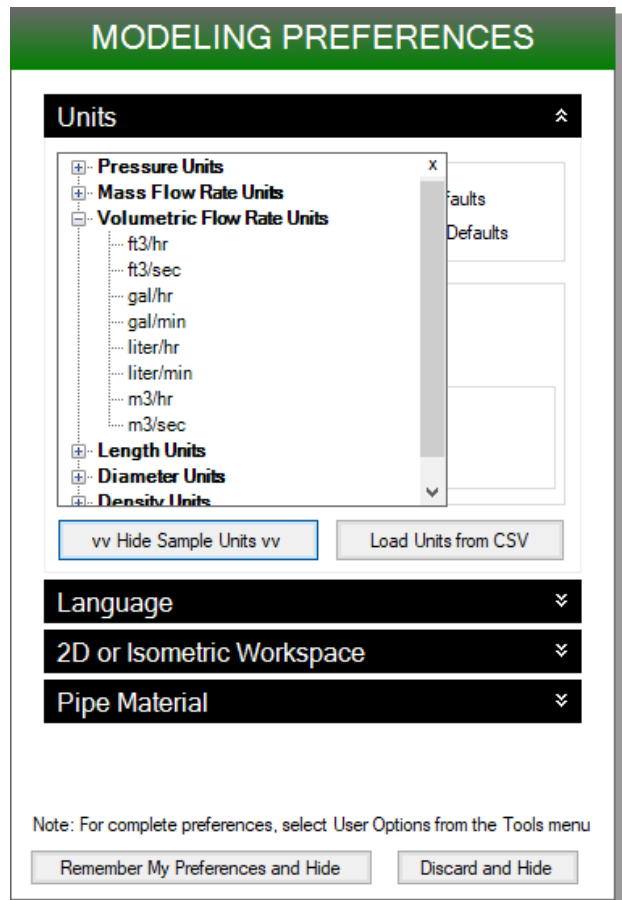
Figure 2.3   Show Sample Units with "Common Only" Selected

The other Filter option is "Common Plus Selected Industries," which will add units from the industries that you select. Once you have finished modifying your Modeling Preferences, click "Remember My Preferences and Hide." Now that your unit preferences are set, click "Start Building Model."

The Workspace window is the initial active (large) window, as seen in Figure 2.4. 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.

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 displayed on the far right. It gives easy access to a variety of features such as the Scenario Manager and viewing pipe and junction properties. The Workspace takes up the rest of the window.
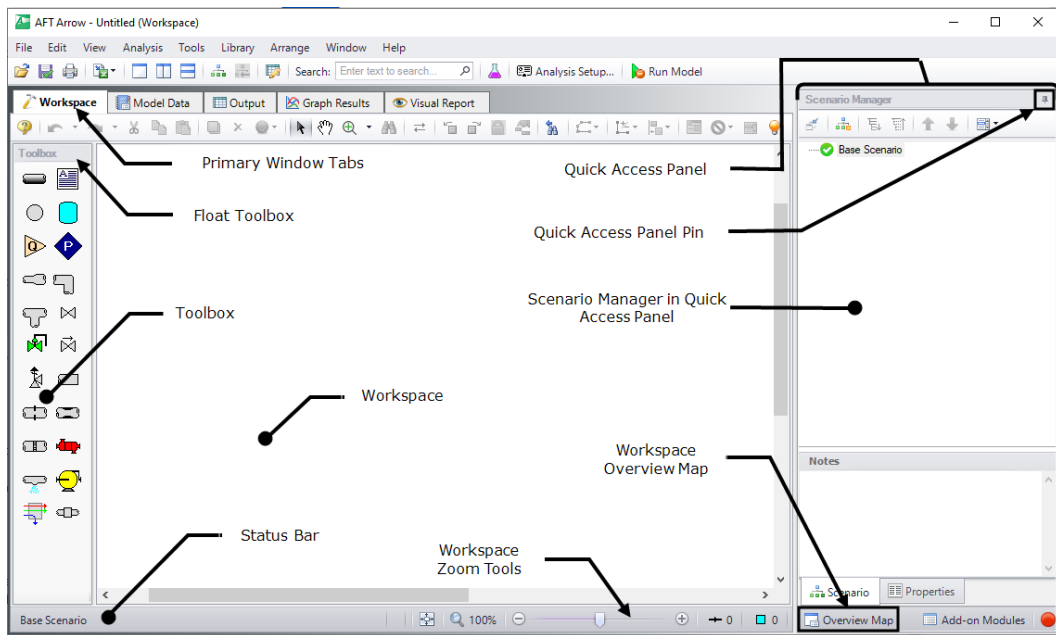
Figure 2.4   The Workspace window is where the model is built. The other four Primary Windows are found on the tabs along the top of the Workspace. The Status Bar shows the model status.

You will build your pipe flow model on the Workspace using the Toolbox tools. At the top of the Toolbox is the Float Toolbar. From here, you can choose the location of the Toolbox in reference to the Workspace by clicking and dragging the Float Toolbar. Below the Float

Toolbar are two drawing tools. The Pipe Drawing tool, on the upper left, is used to draw new pipes on the Workspace. Next to this tool is the Annotation tool. 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 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, the Toolbox tooltips identify the tool's function.

Unpinning the Quick Access Panel

By default, the Quick Access Panel is pinned to the Workspace window so it is always displayed. The Quick Access Panel can be unpinned so it is only displayed when the mouse is moved over the tab displayed on the right edge of the Workspace window. Unpin the Quick Access Panel by clicking on the picture of the pin displayed in the top right corner of the Quick Access Panel (see Figure 2.4). For the remainder of the examples in this guide, the Quick Access Panel will be unpinned.

Step 2. Lay out the model

To lay out the orifice flow model, you will place three junctions on the Workspace. Then you will connect the junctions with pipes.

A. Place a tank

➢ **To start, drag a Tank junction** from the Toolbox and drop it on the Workspace. Figure 2.4a 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 integer greater than zero but less than 100,000.

Editing on the Workspace

Once on the Workspace, junction objects can be moved to new locations and edited using the features on the Edit menu. Cutting, copying, and pasting are all supported. Undo is available for all formatting and arrangement actions on the Workspace.
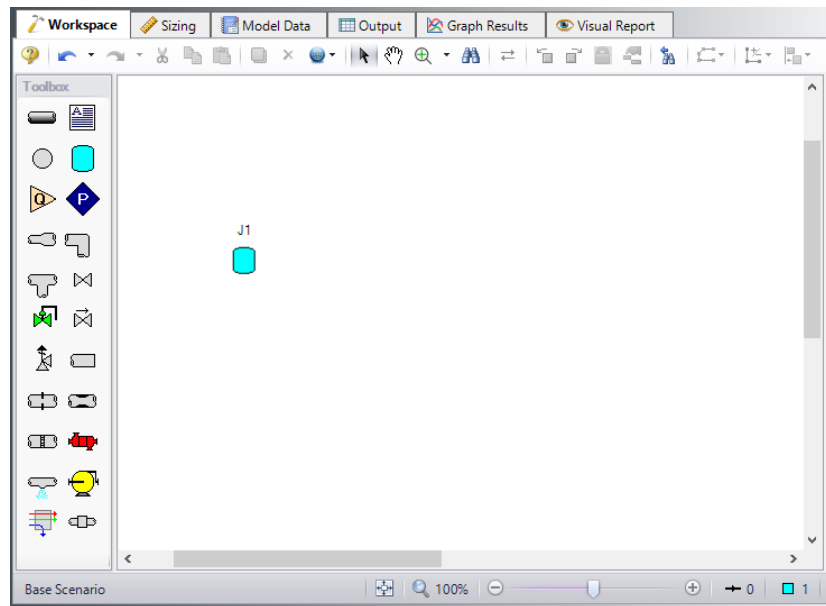
Figure 2.4a   Walk through model with first junction placed

B. 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 choosing Duplicate from the Edit menu.

Drag an Orifice junction from the Toolbox and drop it somewhere to the right of the Tank junction (see Figure 2.4b). Then, either duplicate the first tank, or drag a new Tank junction onto the Workspace. Place the new tank to the right of the Orifice junction (see Figure 2.4b).

Note: The relative location of objects in AFT Arrow is not important. Distances and heights are defined through dialog boxes. These relative object locations on the Workspace establish the connectivity of the objects, but have no bearing on the actual length or elevation relationships. The Isometric Pipe Drawing Mode (see Chapter 3) can be used to visually represent the three-dimensional nature of a system.

If the icons do not line up exactly, the calculations will not be affected, but the model may look better 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 2.4b.
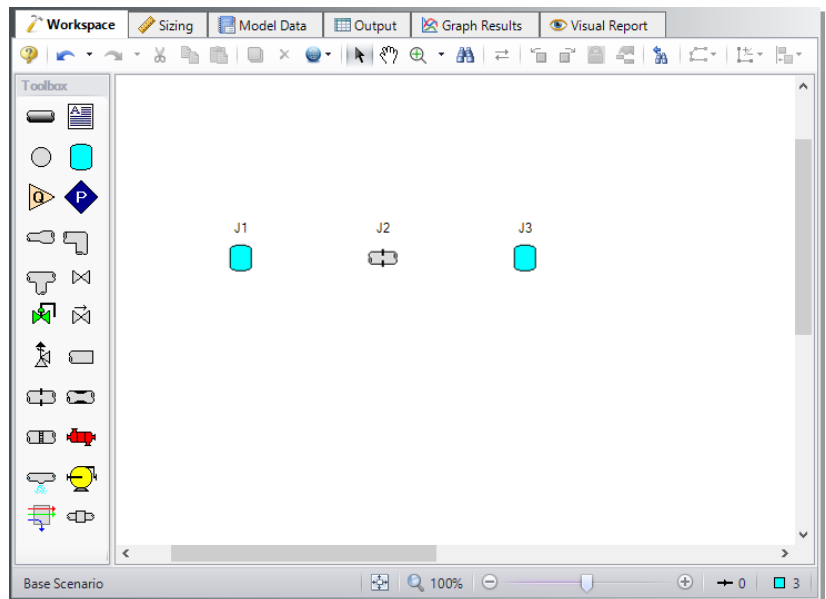
Figure 2.4b   Walk through model with all junctions placed

Before continuing, save the work you have done so far. Choose "Save As" from the File menu and enter a file name (Orifice Flow, perhaps) and AFT Arrow will append the ".aro" extension to the file name.

C. Draw a pipe between J1 and J2

Now that you have three 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 crosshair when you move it over the Workspace. Draw a pipe below the junctions, similar to that shown in Figure 2.4c.

The pipe object on the Workspace has an ID number (P1) that is shown near the center of the pipe.
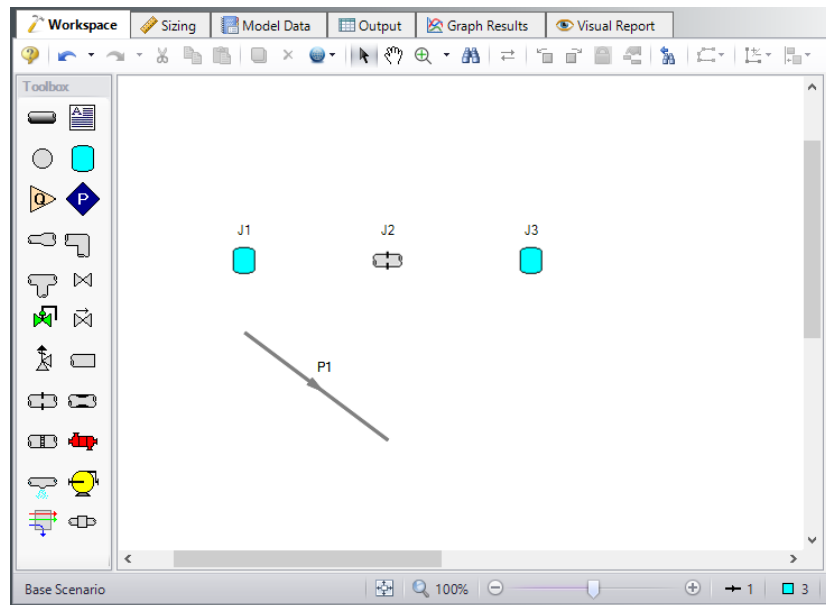
Figure 2.4c   Walk through model with first pipe drawn

➢ **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 icon, then drop it there (see Figure 2.4d). Next, grab the right endpoint of the pipe and stretch the pipe, dragging it until the endpoint terminates within the J2 Orifice icon (see Figure 2.4e).
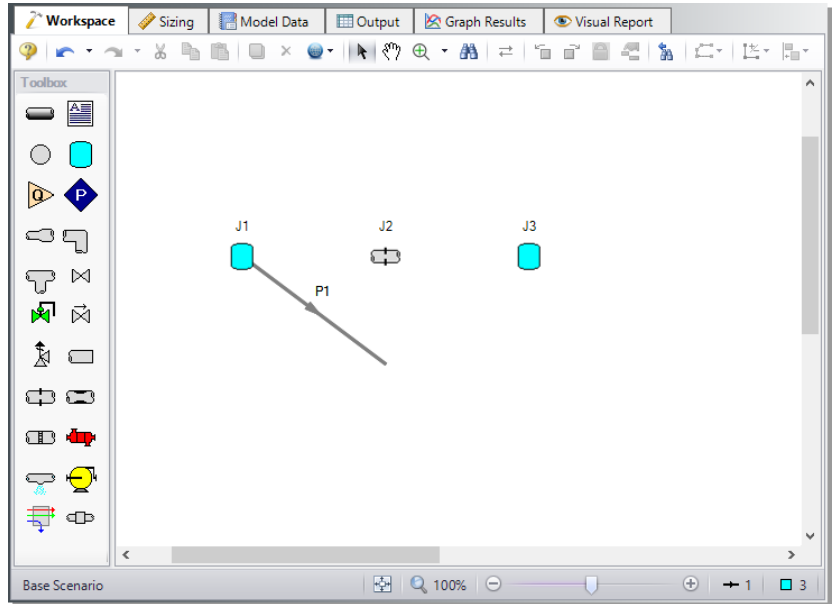
Figure 2.4d   Walk through model with pipe partially connected
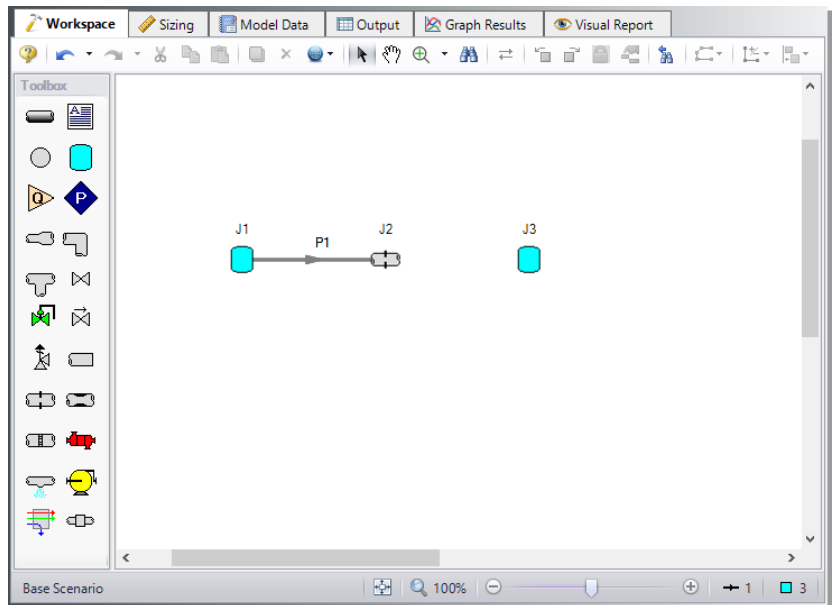
Figure 2.4e   Walk through model with first pipe connected

Reference positive flow direction

There is an arrow on each pipe 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 Toolbar.

In general, the reference positive flow direction is used for reference purposes only and need not be the actual flow direction. However, when used with compressors/fans and certain other junction types, the pipes must be in the correct flow direction because that is how AFT Arrow determines which side is suction and which is discharge. If the reference positive direction is the opposite of that obtained by the Solver, the output will show the flow rate as a negative number.

D. Add the remaining pipe

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 Orifice, then press and hold the left mouse button. Stretch the pipe across to the J3 Tank, and then release the mouse button. Your model should now look similar to Figure 2.4f.

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 save button on the Main 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 and snap to grid. The grid options can be modified through the User Options window from the Tools menu.
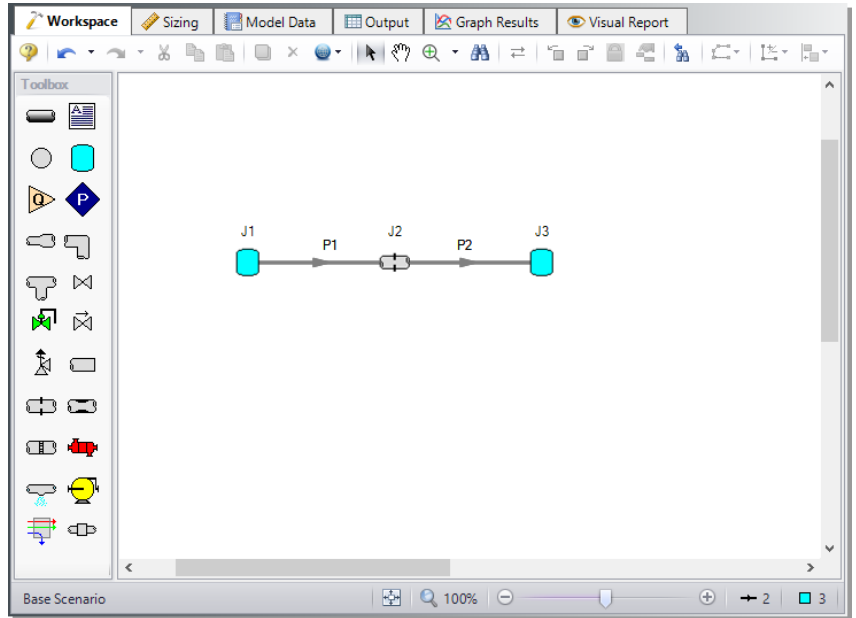
Figure 2.4f   Walk through model with all objects placed

Step 3. Specify Analysis Setup

Analysis Setup is a new feature to AFT Arrow 9 that replaces the Checklist. Select modules, define fluid properties, specify pipe and junction properties, and adjust solution control all from the Analysis Setup window.

➢ **Next, click Analysis Setup on the common toolbar**. This opens the Analysis Setup window (see Figure 2.5). The Analysis Setup window contains six or more groups depending on which modules you may be using. Each group has one or more items. A fully defined group has every item within it defined with all required inputs. Complete every group to run the Solver. A group or item that needs more information will have a red exclamation point to the left of the group or item name. After completing a group, the group icon shows a green circle with a checkmark. After defining all groups, thus completing the Analysis Setup, the Model Status light in the lower right corner turns from red to green.

A. Specify the Modules group

Under the Modules group, click on the Modules item to open the Modules panel, shown in Figure 2.5. Here you can enable and disable add-on modules. These add-on modules include Goal Seek & Control, Extended Time Simulation, and Automated Network Sizing. By default, the model will have all modules disabled. Therefore, make no changes to this panel for this example.
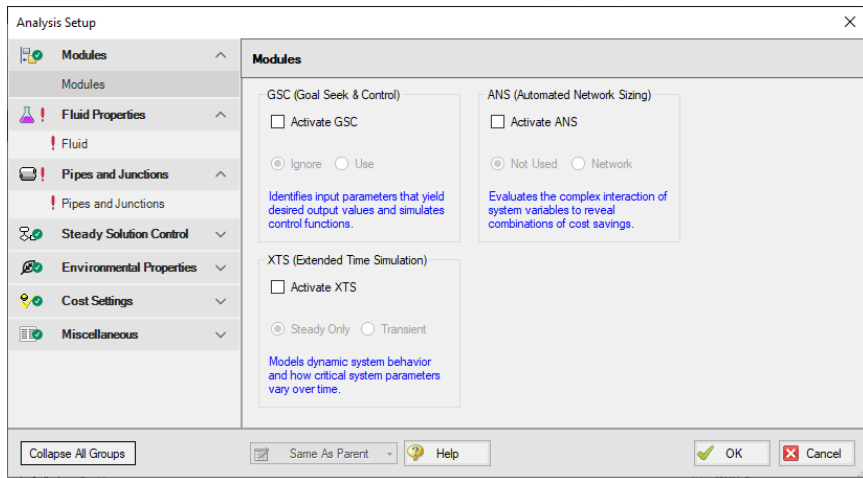
Figure 2.5 **On the Modules panel, you can enable or disable any module**

B. Specify the Fluid Properties group

The Fluid Properties group contains the Fluid panel where the fluid used in the model is defined.

You can select a fluid from the standard AFT Arrow fluid library (AFT Standard), select ASME Steam, select multiple fluids, and/or create mixtures using the NIST REFPROP library (included in AFT Arrow) or the Chempak library (an optional add-on to AFT Arrow), or create custom fluids. Custom fluids are created by selecting Edit Fluids from the AFT Arrow Library menu or by clicking the Edit Fluid List button in the Fluid panel.

➢ **Click on the Fluid item** to open the Fluid panel (see Figure 2.6). For this example, use the AFT Standard fluid library and select "Steam" in

the upper list, then click the "Add to Model" button. Leave the equation of state model as "Redlich-Kwong" and the enthalpy model as "Generalized".
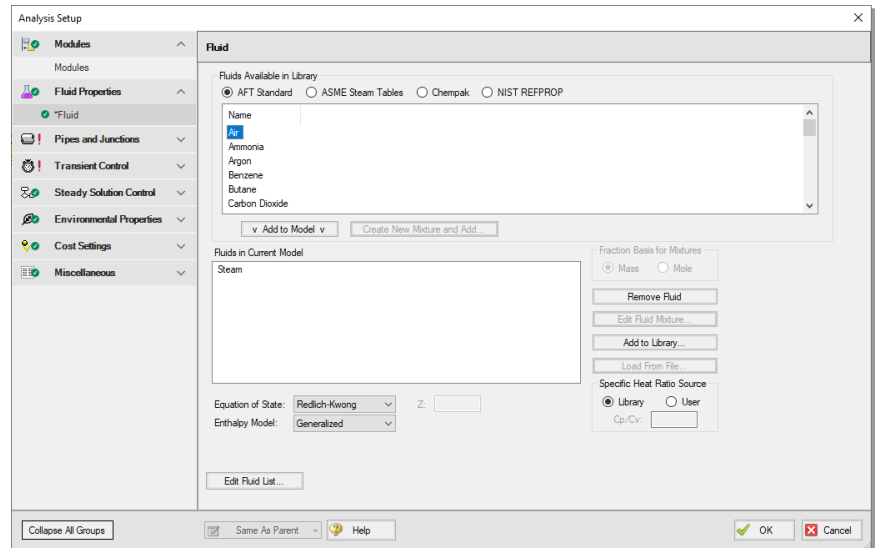
Figure 2.6   Specify the fluid for the analysis in the Fluid panel

C. Specify the Pipes and Junctions group

The Pipes and Junctions group is less straightforward to define compared to the previous two. This group requires full definition for all pipes and junction properties. Click on the Pipes and Junctions item to open the panel shown in Figure 2.7. Here, you will see all the undefined pipes and junctions. By selecting a pipe or junction in the top or middle section, the remaining properties that need definition appears in the bottom section. Select Pipe 1 for example. Pipe Length, Pipe Inner Diameter, and Roughness/Friction needs input. To define these parameters, you can double click on Pipe 1 from top section, or go to the Workspace and open the properties window from there. However before closing the Analysis Setup window, take a look at the final four groups.
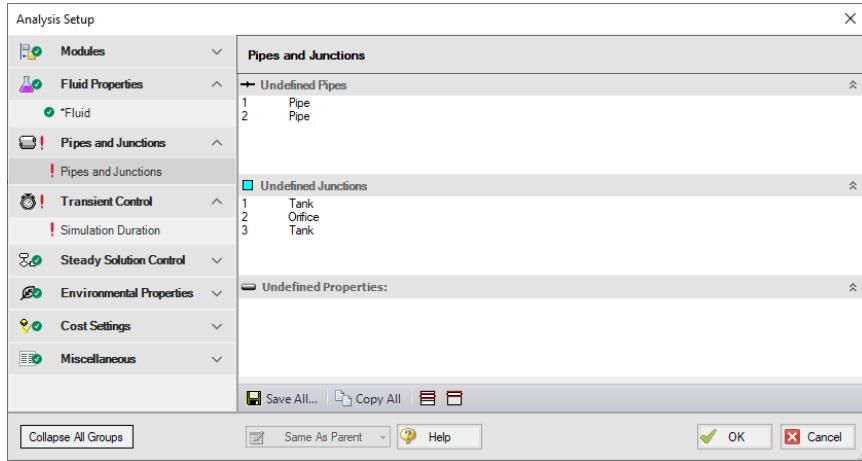
Figure 2.7   The Pipes and Junctions panel lists the remaining undefined objects before the model can run

D. Specify the final four groups

The four remaining groups are Steady Solution Control, Environmental Properties, Cost Settings, and Miscellaneous. In the Steady Solution Control group, you can adjust the tolerance and iteration settings for the solver. On the Environmental Properties panel, you can adjust the ambient pressure, gravitational acceleration, and transition Reynolds Numbers. The Cost Settings panel allows you to select which costs to include as well as define energy costs. The Miscellaneous group has two items to specify heat transfer sectioning, as well as change other advanced parameters. The default selections for all of these groups will suffice to fully define these panels.

➢ **Click OK to save the changes in Analysis Setup** and to return to the Workspace. The remaining undefined Pipes and Junctions group requires input before the model can run. Complete this item by opening the properties window of the objects directly from the Workspace.

Step 4. Define the model components

To see if a pipe or junction has all the required input, check the object's status. Additionally, you can see which remaining objects needs input in

the Undefined Objects list, similar to what was shown in the Pipes and Junctions panel in the Analysis Setup.

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 light bulb icon on the Workspace Toolbar (alternatively, you could choose "Show Object Status" from the View menu). Each time you click the light bulb, "Show Object Status" is toggled on or off.

When "Show Object Status" is on, the ID numbers for all undefined pipes and junctions are displayed in red on the Workspace. Objects that are completely defined have their ID numbers displayed in black. (These colors are configurable through User Options from the Tools menu.)

Because you have not yet defined the pipes and junctions in this sample problem, all the objects' ID numbers will change to red when you turn on "Show Object Status".

Showing undefined objects

Another useful feature is the Undefined Objects list (Figure 2.8). This can be opened from the View menu by clicking on "List Undefined Objects…" or using the List Undefined Objects button on the Workspace Toolbar. Here all objects with incomplete information are listed. Clicking on an undefined pipe or junction will display the property data that is missing. Click the Close button to stop showing this window.
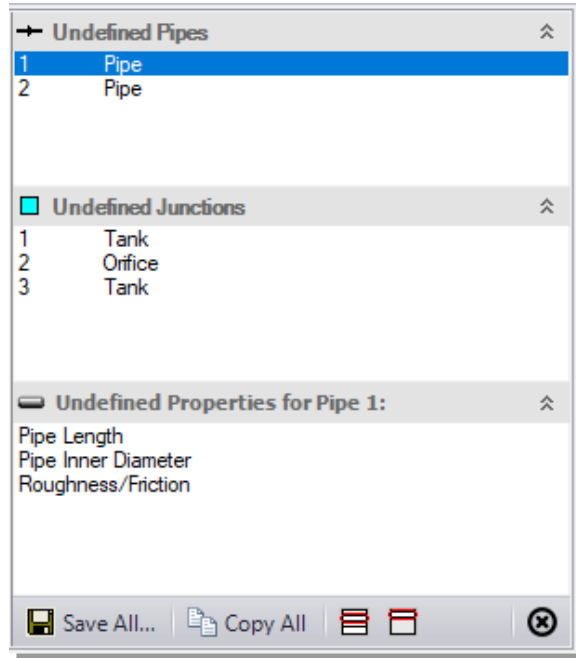
**Figure 2.8   The Undefined Objects list lets you see the undefined
properties for each undefined object**

A. Define Tank J1

➢ **To define the first tank**, open the J1 Tank Properties window (Figure
2.9) by double-clicking on the J1 icon. The pipe table should show you
that Pipe 1 is connected to this tank (click the Loss Coefficients tab to
display the pipe table).

Note: You can also open an object's Properties window by selecting the
object (clicking on it) and then either pressing the Enter key, double-
clicking the object, or clicking the Open Pipe/Junction Window icon on
the Workspace Toolbar.

Figure 2.9   Properties window for Tank J1

➢ **Enter a pressure of 16 barG (1600 kPa(g))**. This tank will represent the supply tank upstream of the orifice. You can assign any unit of pressure found in the adjacent drop-down list of units. Enter a temperature of 260 deg. C and an elevation of 0 meters.

Note: You can choose default units for many parameters (such as meters for length) in the Tools/User Options window under Preferred Units.

You can give the component a name, if desired, by entering it in the Name field at the top of the window. In Figure 2.9, the name of this tank is Inlet Tank. 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 List in the junction's Properties window. The current junction will get the properties from the library component.

The "Copy Data From Jct…" list will show all the junctions of the same type in the model. This will copy the user-selected parameters from a specified existing junction in the model to the current junction.

Using the tabs in the Properties windows

The information in the Pipe and Junction Properties windows is grouped into several categories and placed on separate tabs. Click the tab to bring its information forward. Figure 2.9 is an example of a tank's Properties window.

Click the Loss Coefficients tab in the Properties window of J1 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 for the loss factors to be specified as zero. To change the loss factors later, 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 guess, as well as other junction-specific data.

Each junction has a 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 light blue. The highlight is on by default. You can toggle the highlight off and on by double-clicking anywhere in the window above the tabs or by pressing the F2 key. The highlight feature can also be turned on or off by selecting "Highlight in Pipe and Jct Windows" on the View menu.

➢ **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 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. The inspection window appears, as shown in Figure 2.10.

Inspecting is a faster way of examining the input in an object than opening the Properties window.
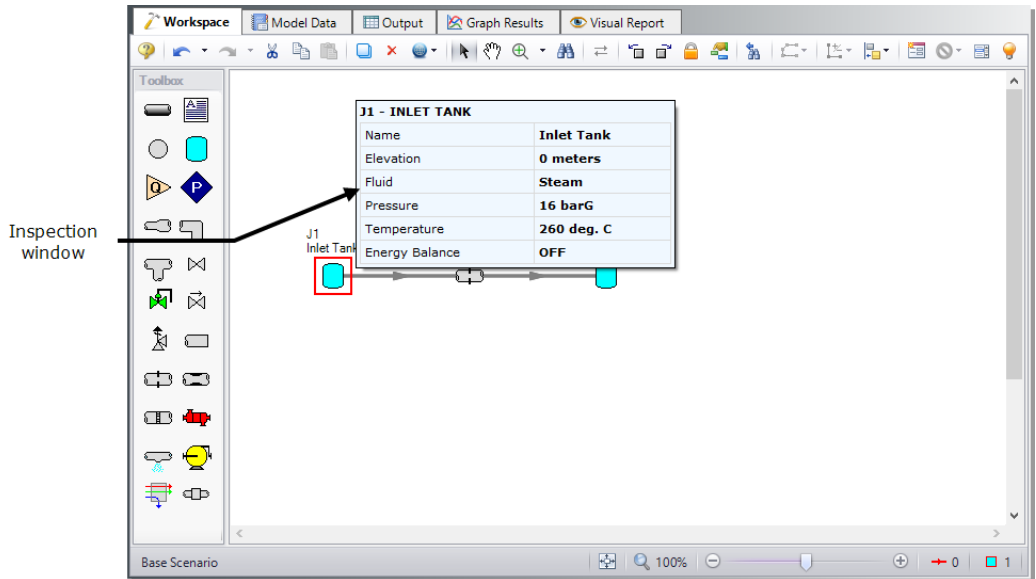
Figure 2.10   Inspecting from the Workspace with the right mouse button

B. Define other junctions

➢ **Next, open the Properties window** for Orifice junction J2. Enter the Orifice Type as User Specified, and the Subsonic Loss Model as K Factor. Enter the orifice area as 20 cm^2, the K factor as 10, and the elevation as 0 meters (see Figure 2.11).

➢ **Open the J3 Tank junction** and change the name to Discharge Tank. Then enter a pressure of 0 barG(kPa(g)), a temperature of 260 deg. C and an elevation of 0 meters.

➢ **Save the model** again before proceeding.
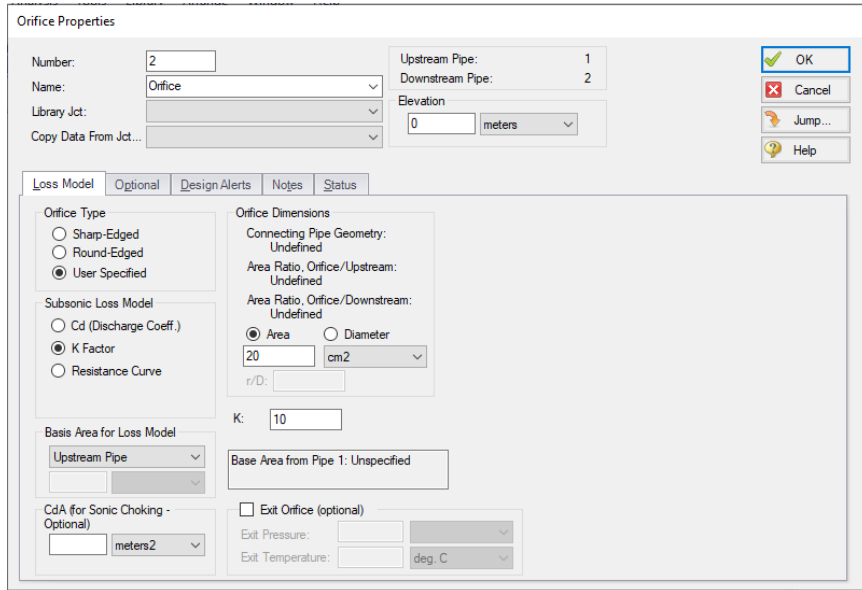
Figure 2.11   Properties window for Orifice J2

C. Define Pipe P1

Data for pipes and junctions can be entered in any order. In this example, the junctions were done first. The next step is to define all the pipes. To open the Pipe Properties window, double-click the pipe object on the Workspace.

➢ **First, open the Pipe Properties window** for Pipe P1 (see Figure 2.12). For Pipe P1 enter Steel - ANSI, 2 inch, STD (schedule 40), Standard friction model, and a length of 3 meters.

The Pipe Properties window

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

The Inspect feature can be accessed not only from pipes and junctions located on the Workspace, but also from within the Properties window of each pipe (and certain junctions). This is helpful when you want to quickly check the properties of objects that connect to a pipe or junction whose Properties window you already have open.

To Inspect a junction connected to a given pipe, position the mouse pointer on the connected junction's ID number in that pipe's Properties window (located at the top right of the Pipe Properties window) and hold down the right mouse button. This process can be repeated for any junctions that state the upstream and downstream pipe in the junction's Properties window by holding the right mouse button on the pipe's ID number.
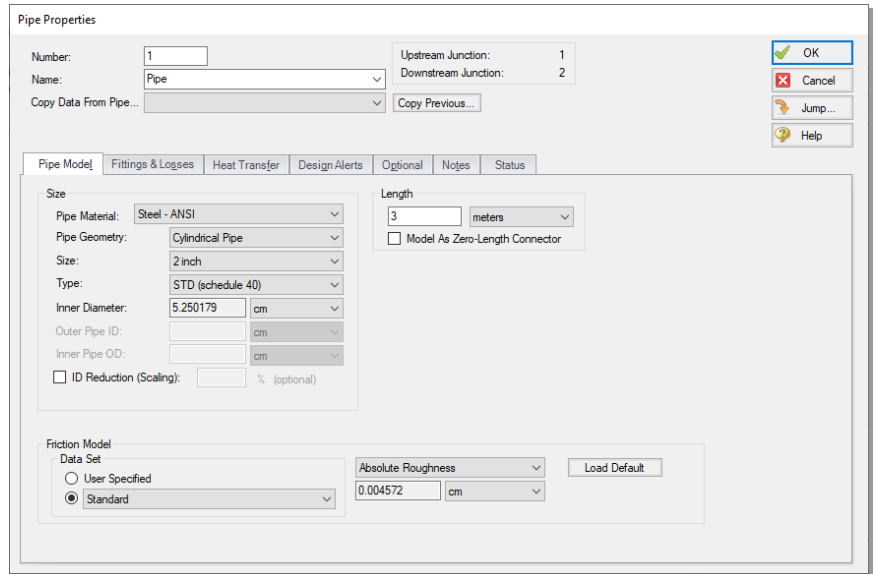
Figure 2.12   Properties window for Pipe P1

By double-clicking the connected junction number from a Pipe's Properties window, you can jump directly to the junction's Properties window. Or you can click the Jump button to jump to any other part of your model.

D. Define Pipe P2

➢ **Open the Properties window for Pipe 2** and specify Steel - ANSI, 2 inch, STD (schedule 40), Standard friction model, and a length of 6 meters. You will note that the pipe size and type is remembered from the previous pipe (this can be configured in the User Options window under Pipes and Junctions in the Parameters section). You can also

choose the "Copy Data From Pipe" dropdown and select the parameters from any other pipe to apply to this pipe.

E. Check pipes and junctions

➢ **Check if all the pipes and junctions are defined**. If all the panels in the Analysis Setup have their required inputs, the Status Light on the Quick Access Panel will be green. If not, view the Undefined Objects list from the Analysis Setup, or turn on Show Object Status from the View menu, and open each undefined pipe and junction. The Status tab on each Properties window will indicate the missing information.

F. Review 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 2.13. To change to this window, you can click on the Model Data tab, select it from the Window menu, or press Ctrl + M. The Model Data window gives you a text-based perspective of your model. Selections can be copied to the clipboard and transferred into other Windows programs, saved to a formatted file, printed to an Adobe™ PDF, or printed out for review.

Data is displayed in three sections. The top is called the General data section, the middle is the Pipe data section and the bottom is the Junction data section. Each section is collapsible using the buttons at the top left of the section. Further, each section can be resized.

The Model Data window allows access to all Properties windows by double-clicking on any input parameter column in the row of the pipe or junction you want to access. You may want to try this right now.

General data section

Pipe data section

Junction data section
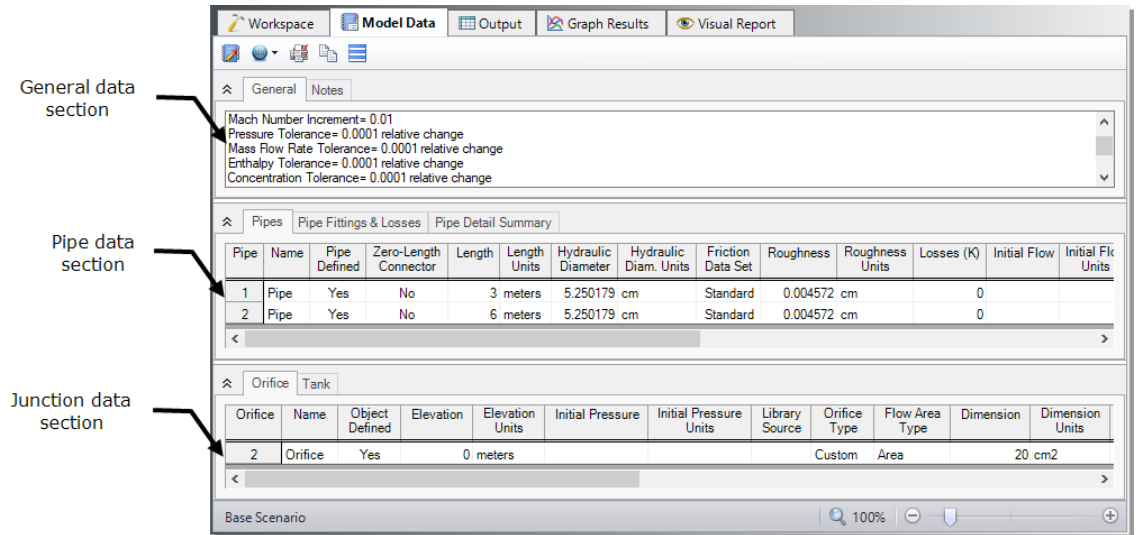
Figure 2.13   The Model Data window shows all input in text form

Step 5. Run the Solver

➢ **Choose "Run Model"** from the Run Model icon on the Main Toolbar or the Analysis menu. During execution, the Solution Progress window displays (see Figure 2.14). You can use this window to pause or cancel the Solver's activity. When the solution is complete, click the Output button and the text-based Output window will appear (see Figure 2.15). 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.
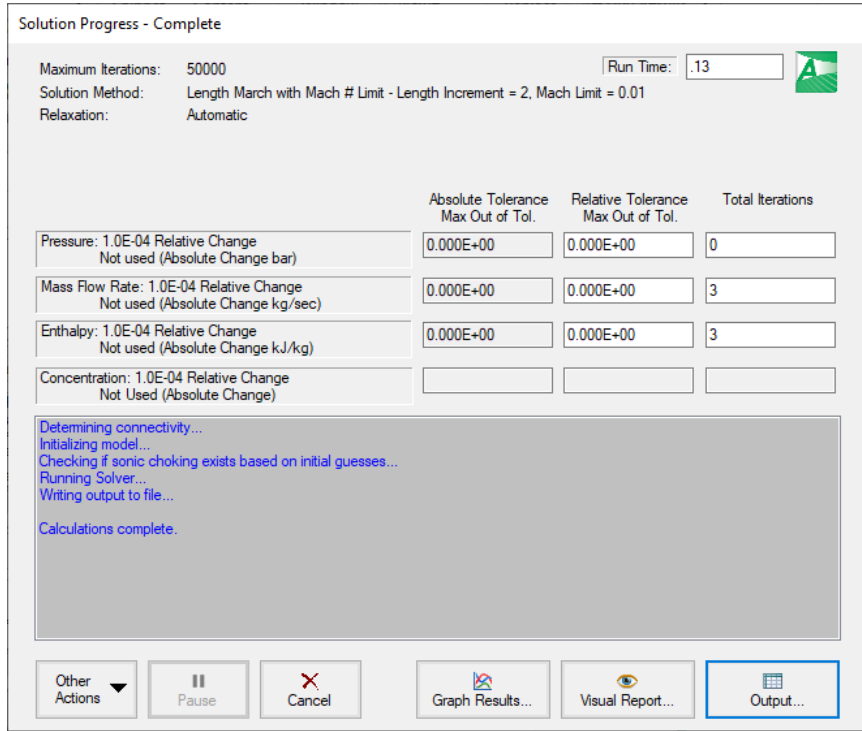
Figure 2.14   The Solution Progress window displays the convergence progress

Step 6. Review the output

The Output window (Figure 2.15) is similar in structure to the Model Data window. Three areas are shown, and you can expand or collapse each area by dragging the boundary between the areas up or down, or by clicking the arrow beside the area label. The items displayed in the tables are those items you choose in the Output Control window.
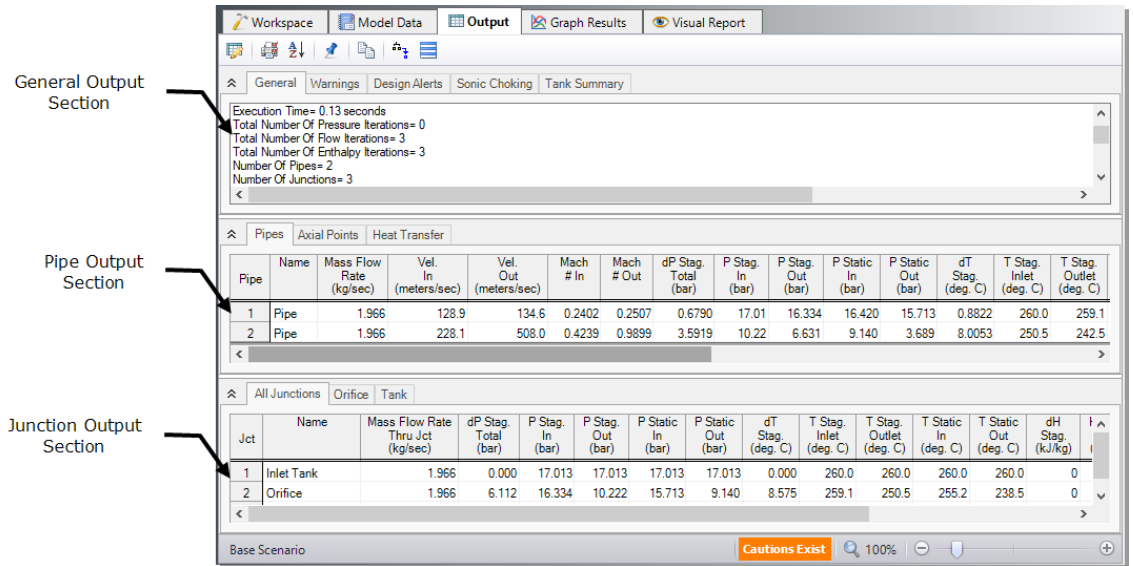
Figure 2.15   The Output window displays output in text form

This system chokes at the discharge tank. When a system chokes, Arrow will display the "Sonic Choking" tab in the General Output section (see Figure 2.16). This tab will appear when sonic choking occurs in the system, and will contain 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 1.97 kg/sec. The sonic pressure at the exit is 6.63 bar (663 kPa) (i.e., stagnation pressure at Pipe 2 outlet). The output in the junction data portion of the Output window shows the Sonic Area at the orifice is 8.89 cm^2, and the Sonic Area at the discharge tank is 21.65 cm^2. The sonic area is equal to the pipe physical area of the 2 inch, standard schedule pipe since the system chokes at the discharge tank, and the connecting pipe, P2, has no restriction at the end.

Figure 2.16   The Output window displays the Sonic Choking tab whenever sonic choking occurs in the system

Step 7. Customize the output

The Output Control window (Figure 2.17) allows you to select the specific output parameters you want in your output. You also can choose the units for the output. If you do not change any of the Output Control settings, default Output Control parameters and a default title are assigned.

➢ **Select Output Control** from the Tools menu or the Main Toolbar to open the Output Control window (Figure 2.17). On the Display Parameters tab, select the Pipes button. The list of selected pipe output parameters will be displayed on the right hand side. If you selected the default AFT Arrow Output Control, the pipe results in the Pipes table will show mass flow rate in the second column with units of kg/sec. Change the units for Mass Flow Rate by selecting kg/hr (kilograms per hour) from the unit list shown beside the parameter.
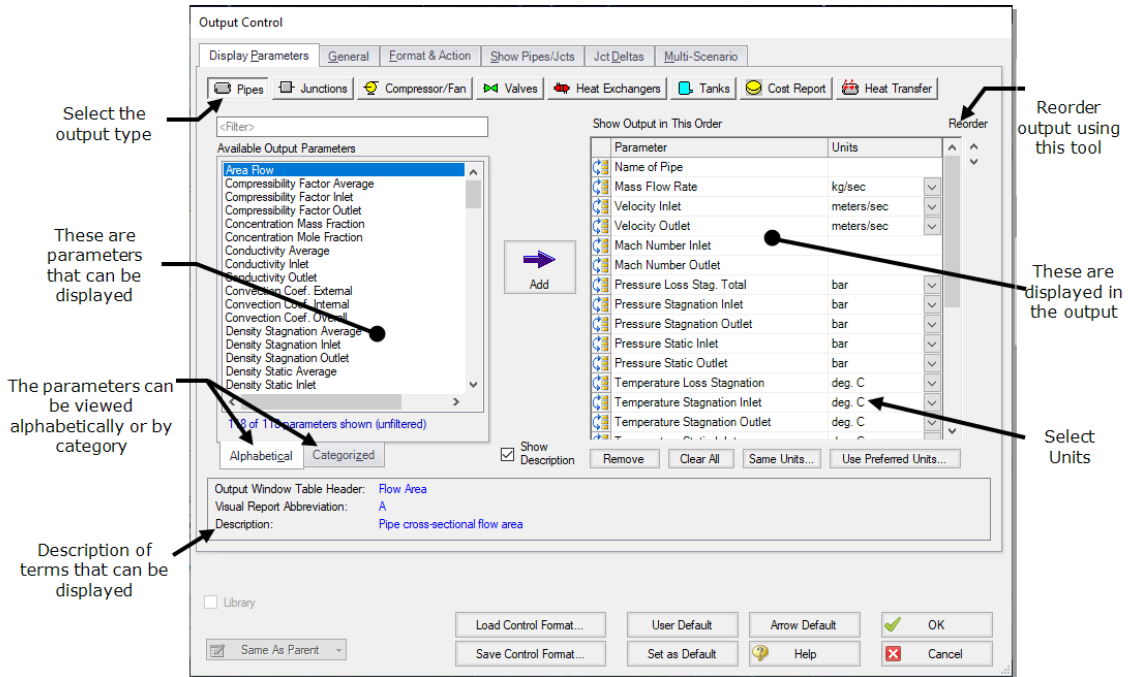
Figure 2.17   The Output Control window lets you customize the output

➢ **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 kg/hr.

➢ **Select Output Control from the Tools menu** one more time. Select the Display Parameters tab, and select the Pipes button. The Reorder scroll bar on the far right allows you to reorder the sequence of parameters that will be displayed in the output. You can also click on the reorder symbol on the left side of each row to manually drag-and-drop the rows to change the parameter order (see Figure 2.17).

➢ **Select the Velocity Inlet parameter** and use the Reorder scroll bar or drag-and-drop to move it up to the top of the parameter list.

➢ **Click OK to display the changes** to the current results. You will see in the pipe results in the Pipes table that the first column now contains Vel. In (Velocity Inlet), the second column now contains the pipe name, and the third column contains the mass flow rate. The Output Control

window allows you to specify the parameters, units and order of the displayed parameters that you want to see in the 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 in the Pipes table. This will open a window in which you can change the units once again if you prefer (see Figure 2.18). These changes are extended to the Output Control parameter data you have previously set.
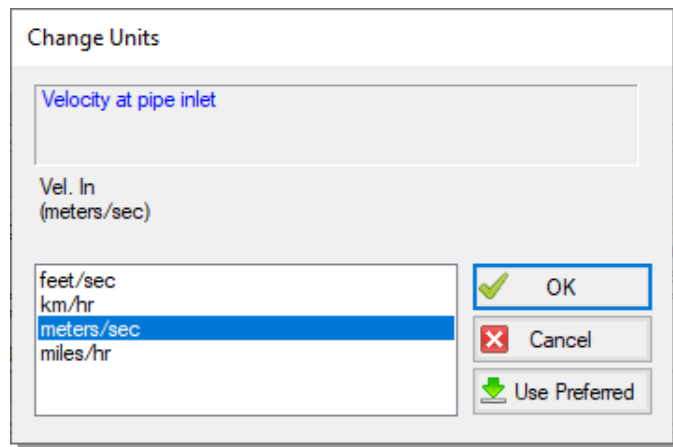
Figure 2.18   The Change Units window is opened from the Output window tables by double-clicking the column header

Step 8. View the Visual Report

➢ **Change to the Visual Report window** by clicking on the Visual Report tab, choosing it from the Window menu, or pressing Ctrl + I. This window allows you to integrate your text results with the graphic layout of your pipe network.

➢ **The Visual Report Control window** will open by default when you first open the Visual Report window (Figure 2.19). You can also click the Visual Report Control button on the Visual Report Toolbar (or Tools menu) to open the Visual Report Control window. Select Velocity Inlet in the Pipe Results area and Pressure Static Inlet in the Junction Results

area. Click the Show button. The Visual Report window graphic is generated (see Figure 2.20).

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. You can also use the Visual Report Control window to display units in a legend to increase the clarity of the display. These adjustments have already been done in Figure 2.20. 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.
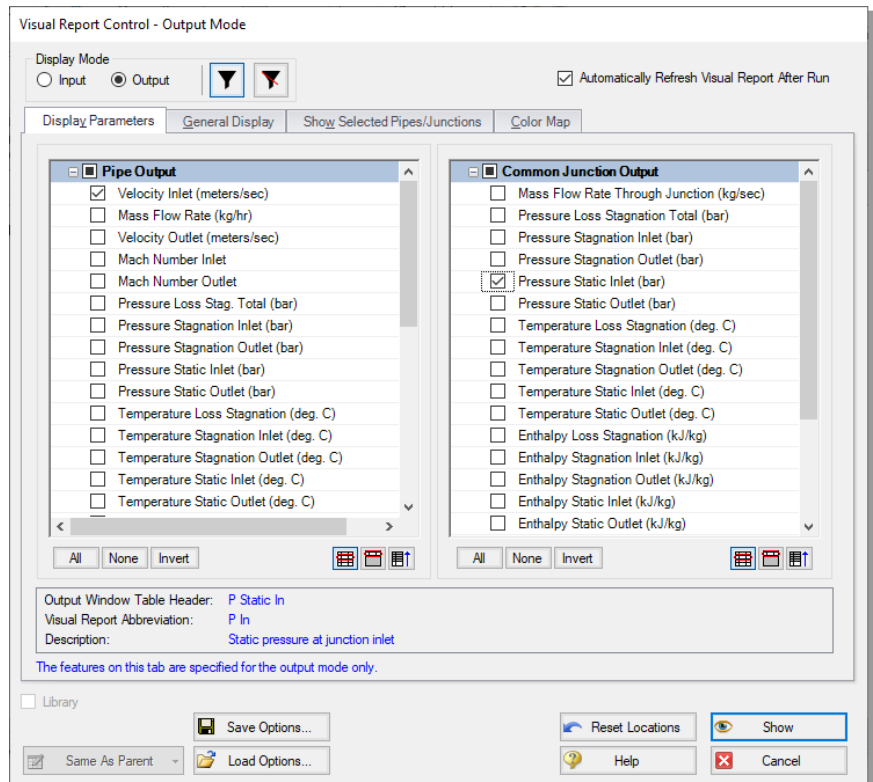
Figure 2.19   The Visual Report Control window selects content for the Visual Report window
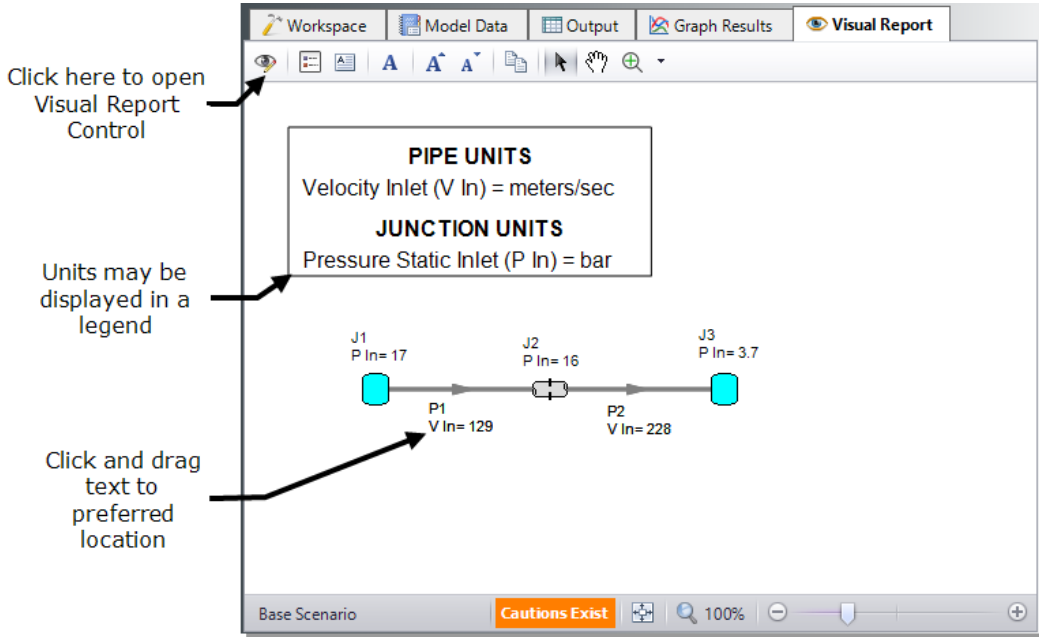
Figure 2.20   The Visual Report integrates results with model layout

Step 9. View the Graph Results

➢ **Change to the Graph Results window** by clicking on the Graph Results tab or choosing it from the menu. The Graph Results window offers full-featured Windows plot preparation.

AFT Arrow's Graph Guide, accessed by clicking on the "What Would You Like to Do?" button located at the top right of the Graph area, provides assistance by guiding you through the creation of a "Quick and Simple" graph, or an "Advanced" graph (Figure Figure 2.21). You can create a graph by following the prompts on the Graph Guide, by manually specifying the graphing parameters on the Graph Control tab on the Quick Access Panel, or from the "Select Graph Parameters" icon located in the top left corner of the Graph Results toolbar. For the remainder of this Quick Start Guide, the Graph Guide will be hidden, but keep in mind that you can use it whenever you would like assistance in creating a graph.

Figure 2.22 shows the Graph Results window with the Graph Control tab enabled on the Quick Access Panel.
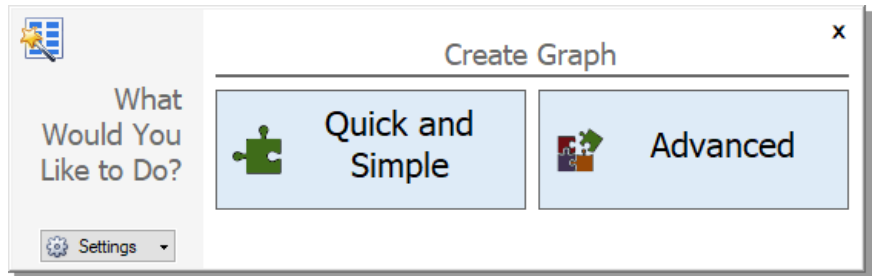
Figure 2.21   The Graph Guide can be toggled on and off by clicking the "What Would You Like to Do?" button on the Graph Results window
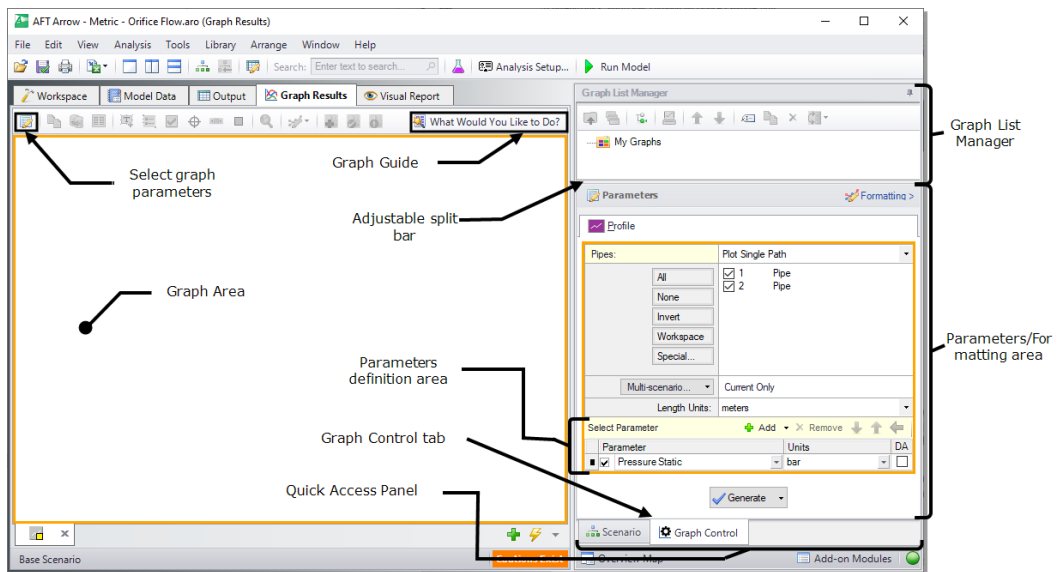
Figure 2.22   Graph Results window offers full-featured graphing

AFT Arrow gives you the ability to create "stacked graphs". These are graphs that are displayed on top of each other with the same X-axis but with different parameters on the Y-axis. This feature is very helpful when you want to look the behavior of multiple hydraulic parameters in

the same location of your model simultaneously without having to create completely separate graphs.

In this example, you will create stacked graphs of the pressure and velocity along the pipeline.

➢ **Click the Profile tab** on the Quick Access Panel. In the Pipes selection list, click the All button to select all the pipes.

➢ **For the Length Units**, select Meters.

➢ **From the Parameters definition area**, select "Pressure Static" and specify units of barG.

➢ **In order to add a stacked graph**, click the "Add" button, which is the green "+" icon next to "Select Parameter." A new row under the Parameters definition area will appear.

➢ **Choose "Velocity" in this new parameter row**, and select units of meters/sec.

➢ **Click the Generate button** to create the stacked pressure and velocity graphs.

➢ **To format the legend font size**, right-click on each legend and use the slider bar to decrease the font size to 12 for both graphs. Drag the static pressure graph legend to the upper right corner of the graph to improve visibility of the graphed data.

➢ **To format each axis font size**, right-click on each axis title and use the slider bar to decrease the size until the font size on each axis appears as you like.

Figure 2.23 shows the input in the Parameters/Formatting area on the Quick Access Panel.

Figure 2.24 shows the stacked graphs detailing the static pressure and velocity along the entire pipeline modeled in this simple system. Note that the total static pressure loss across the orifice is approximately 6.6 bar.

The Add button adds another parameter to your graph
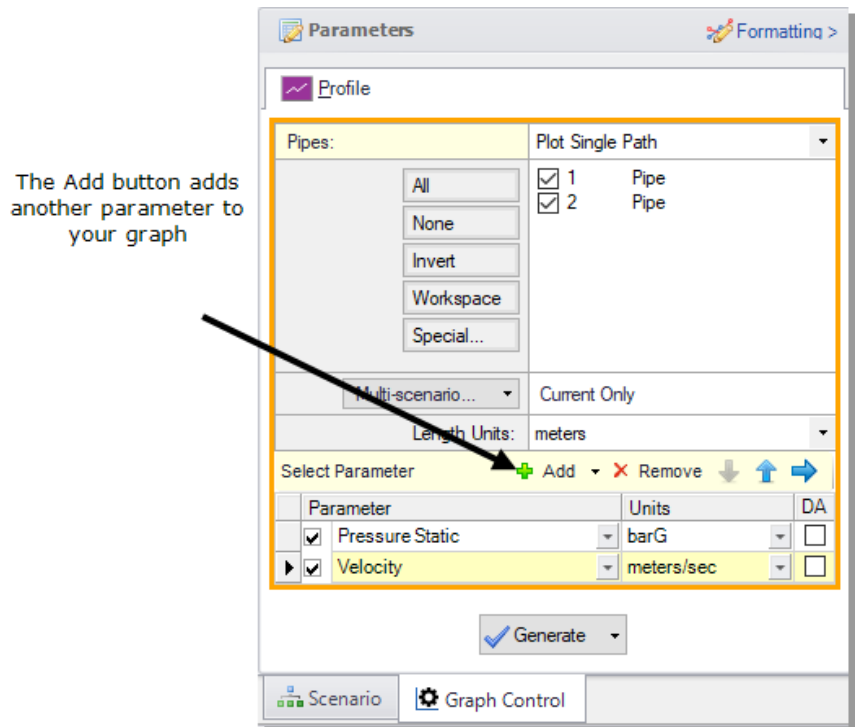
Figure 2.23   **The Graph Control tab on the Quick Access Panel allows you to specify the graph parameters you want to graph in the Parameters/Formatting area**
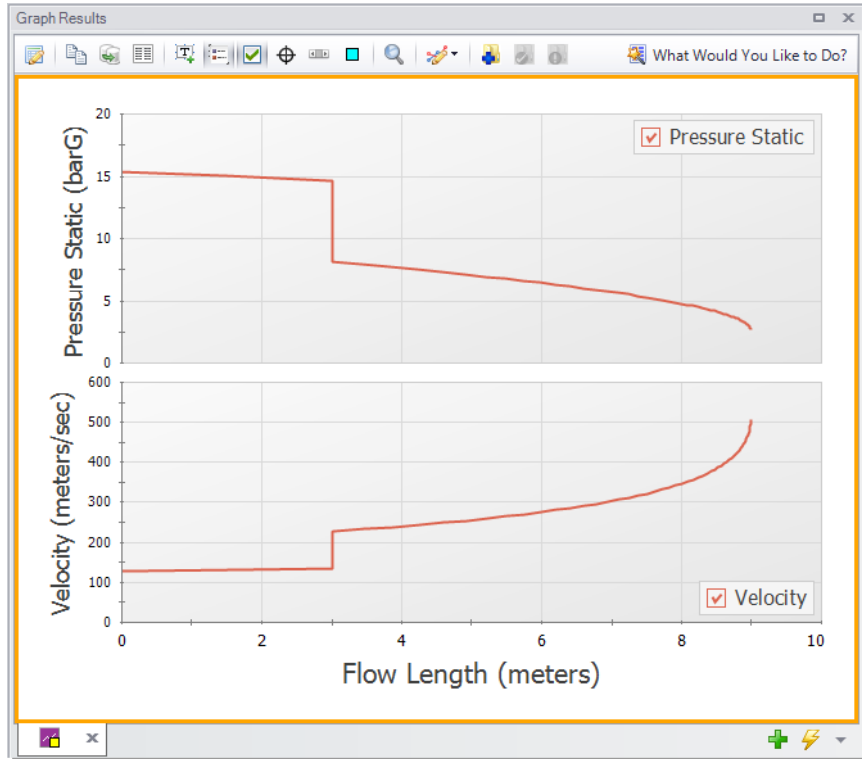
Figure 2.24   The Graph Results window offers full-featured graphing

The graph colors, fonts, and other elements can be modified using the Formatting area on the Graph Control tab on the Quick Access Panel. 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 a file or copied to the clipboard.

Step 10. Modify the system

So far in this example, there has been no input for "CdA for Sonic Choking" on the Orifice Properties window. This input is optional, but without a CdA input, sonic choking cannot be calculated at this location, and choking would not occur at the orifice. Sonic choking can occur at a different location, as was seen at Tank Junction J3.

To see the effects of CdA, enter a CdA that is 20% higher than the sonic area, and run the Arrow model again. The previous sonic area at J2 was 8.89 cm^2, and the new value should be 10.668 cm^2 (i.e., 20% higher). How do the results change? What happens if you set the optional orifice CdA to a value 20% lower than the sonic area (7.112 cm^2)?

Note: This is a situation where a user could create two new scenarios using the Scenario Manager to examine several "what-if" situations, without disturbing the basic model. See Chapter 3 for an example that illustrates how to use the Scenario Manager.

When the orifice CdA is set to a value higher than the sonic area, it has no effect on the model. However, setting the orifice CdA to a value lower than the sonic area will result in sonic choking at that location.

Here, a value 20% lower than the sonic area causes the flow rate to decrease from 1.97 to 1.59 kg/sec, and the sonic discharge pressure to decrease to 5.37 bar (537 kPa). For this system, lowering the orifice CdA to a value 20% less than the sonic area also causes the system to choke in two places rather than just one. This can be seen on the Sonic Choking tab in the General Output window.

Conclusion

You have now used AFT Arrow's five Primary Windows to build and analyze a simple model.

Compressed Air System Analysis Example

This example demonstrates how to determine the range of outlet temperatures for a compressed air system, given a range of numerous design assumptions.

Topics covered

This example will cover the following topics:

- Entering heat transfer data for piping

- Modeling compressors with curve data

- Modeling heat of compression at compressors

- Modeling junction losses with a resistance curve

- Using scenarios

- Using the Global Pipe Edit feature

Required knowledge

This example assumes that the user has some familiarity with AFT Arrow, such as placing junctions, connecting pipes, and entering pipe and junction properties. Refer to the Flow Through an Orifice Example in Chapter 2 for more information on these topics.

Model file

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

- *Metric - Compressed Air System.aro* - AFT Arrow model file

This example is provided in English units as *US - Compressed Air System.aro* - AFT Arrow model file.

Problem statement

In this example, four machine tools are supplied air for operations. The machine tools are sensitive to temperature, but the manufacturer says they can compensate for this if they know the extremes of the delivery temperature the tools will see.

The air is taken from outside the building (P = 0 barG), and the air temperature outside can vary from -18 to 45 deg. C. The temperature inside the building ranges from 21 to 24 deg. C. The air velocity ranges from 1.5 to 25 km/hr.

The compressor used to drive the system has the following stagnation pressure rise characteristics: 83 kPa at 0 kg/sec, 69 kPa at 0.25 kg/sec, and 41 kPa at 0.5 kg/sec. The compressor efficiency is not known with certainty, but it is expected to be between 80% and 90%.

The nozzles at the tools have a pressure drop of 55 kPa at 0.1 kg/sec. The nozzles discharge to atmospheric pressure.

The pipes in the system are uninsulated steel. The pipe at the compressor inlet is heavily insulated (adiabatic). All of the components in the system are at an elevation of 0 meters.

Determine the temperature extremes of this system, so the machine tools manufacturer can compensate for the tool's temperature sensitivity.

Step 1. Start AFT Arrow

From the Start Menu, choose AFT Products and AFT Arrow.

Step 2. Specify fluid properties

1. Open Analysis Setup by selecting Analysis Setup from the common toolbar or the Analysis menu.

2. Under the Fluid Properties group open the Fluid panel if not already open. Select the AFT Standard library and then select "Air" in the Fluids Available in Library list.

3. Click "Add to Model" to select air for use in this model

4. Use the default "Redlich-Kwong" for the equation of state and "Generalized" for the enthalpy model

Step 3. Build the model

A. Place the pipes and junctions

This completes the first two groups of the Analysis Setup. The Pipes and Junctions group remains undefined. To complete this group, begin by assembling the model on the Workspace as shown in Figure 3.1.
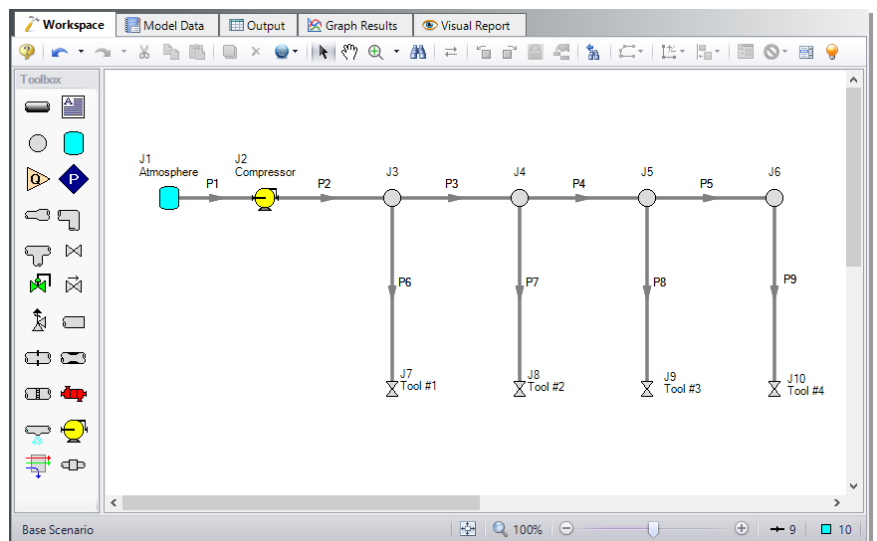

Figure 3.1   **Layout of pipe system for Compressed Air System Example**

B. Enter the pipe data

The system is in place, but now you need to enter the input data for the pipes and junctions. Double-click each pipe object, and enter the following data in the Properties window:

All of the pipes are Steel - ANSI, with standard roughness and the following data:

Pipe	Length (meters)	Nominal Size	Type
P1	0.3	2 inch	Schedule 40
P2 - P5	8	2 inch	Schedule 40
P6 - P9	3	1 inch	Schedule 40

1. Because the pipe at the compressor inlet (P1) is very well insulated for all of the cases you will analyze, it can be considered adiabatic. Open the Properties window for pipe P1 and select the Heat Transfer tab. Ensure that the Heat Transfer Model is set to "Adiabatic (Perfectly Insulated)", then click OK.

2. For the remaining uninsulated pipes (P2 - P9), you will need to set the pipe heat transfer properties on the Heat Transfer tab on the Pipe Properties window.

   AFT Arrow has a feature that can be used to change the heat transfer properties for multiple pipes at the same time. This feature is called Global Pipe Edit.

Open the Global Pipe Edit window from the Edit menu. Click "Select All" to select all of the pipes shown in the Pipe List. Then click on the check box beside Pipe 1 to deselect it. Since Pipe 1 has different properties than the other pipes (it is adiabatic), you should not include it in this global change. The Global Pipe Edit window should appear as shown in Figure 3.2.
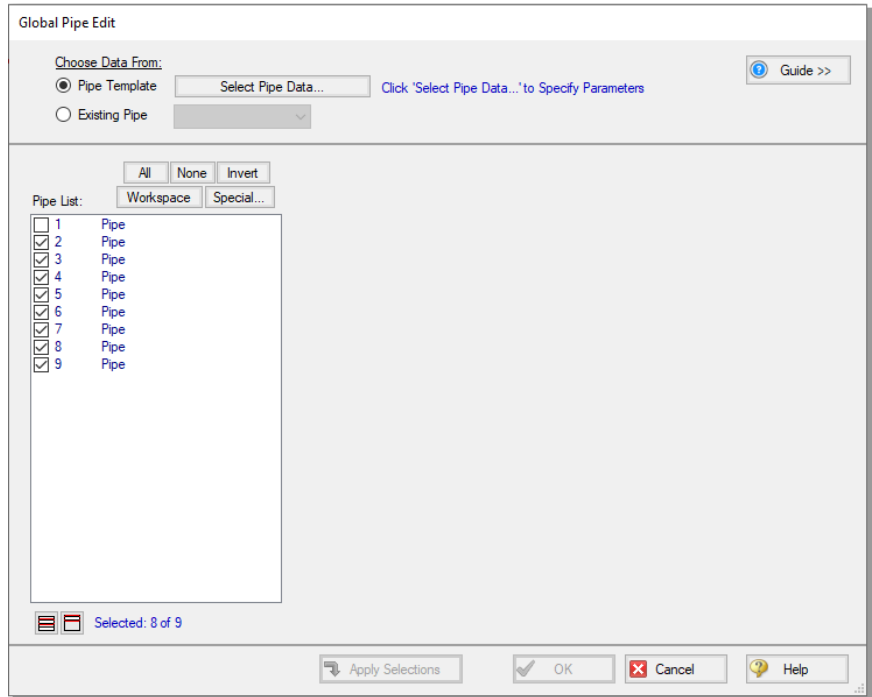
Figure 3.2   Global Pipe Edit window showing the pipes selected to be included in the global edit

Click the Select Pipe Data button. This will open a Pipe Properties window that will be used to enter the pipe parameters we wish to change on all of the selected pipes.

On the Pipe Model tab, set the Pipe Material to Steel - ANSI. No other changes are necessary on this tab.

Click on the Heat Transfer tab. Select "Convective Heat Transfer" for the Heat Transfer Model, and enter 24 deg. C for the Ambient Temperature and 1.5 km/hr for the Fluid Velocity with Air as the fluid. Also, ensure that the Gnielinski correlation is selected under the Convection Coefficient column in the Fluid Internal row, and that Forced (Churchill-Bernstein) is chosen in the External row. The Pipe Properties window should appear as shown in Figure 3.3. Select OK.
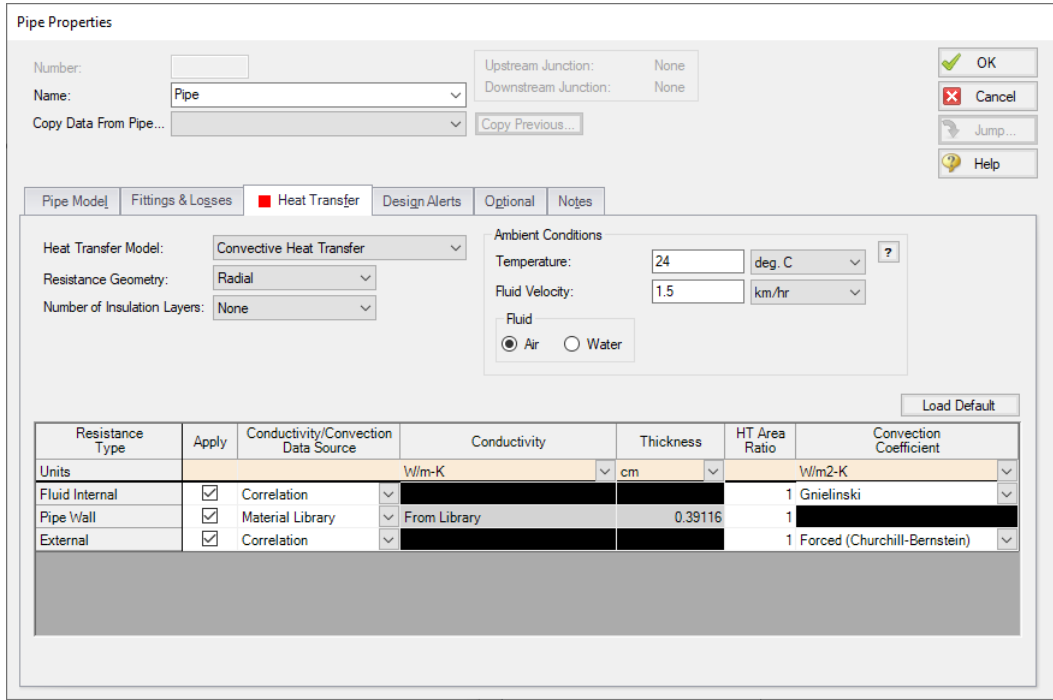
Figure 3.3   The Heat Transfer tab is where you specify heat transfer properties for the pipe

The Global Pipe Edit window will now display a list of all the parameters that may be applied to the selected pipes. The parameters are categorized as they are displayed on the tabs on the Pipe Properties window. For this example, you want to select all of the parameters listed in the Heat Transfer category (see Figure 3.4). Since these items are the only things selected in the parameter list, they are the only parameters that will be changed in the pipes.

Select the Apply Selections button, acknowledge the "changes complete" message, and then select the OK button to close the Global Pipe Edit window.

The heat transfer data for Pipes P2 - P9 have all been updated to reflect the new values. Open the Properties window for one of the pipes to verify that the changes were made.

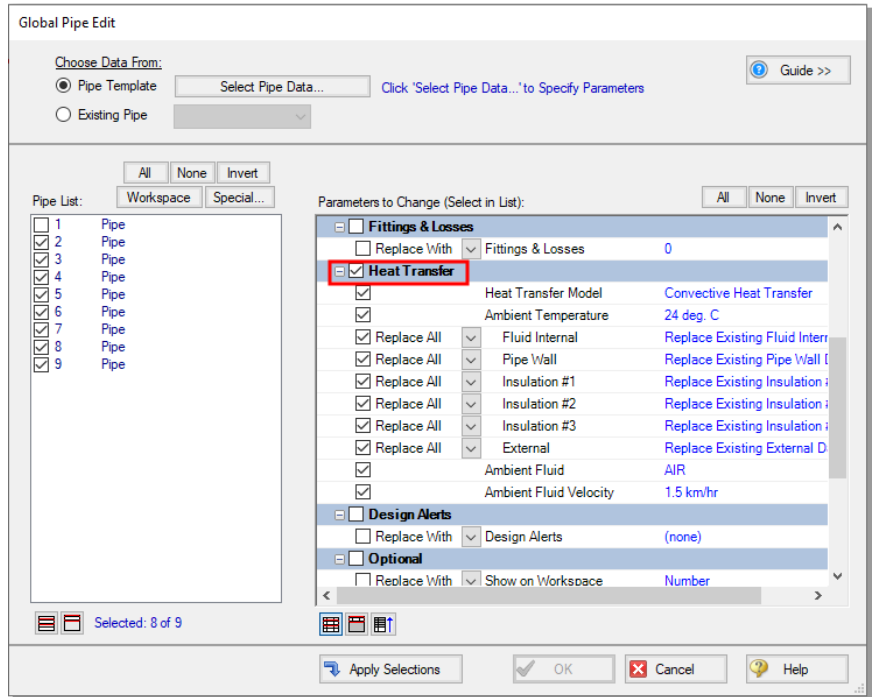**Figure 3.4**   **Global Pipe Edit window showing the parameters to be applied to all pipes selected in the Pipe List**

C. Enter the junction data

J1 Atmosphere

1. Elevation = 0 meters

2. Pressure = 0 barG

3. Temperature = 45 deg. C

J2 Compressor

1. Elevation = 0 meters

2. Select Centrifugal Compressor and set the Compressor Model to Compressor Curve

3. Choose the "Added Pressure" as Stagnation

4. Set the "Compression Process Thermodynamics" to "Determine From Efficiency Data"

5. Select the Enter Curve Data button. Enter the Compressor Curve pressure and efficiency data from the following table into the Raw Data table in the Compressor Configuration window that is shown in Figure 3.5

m' (kg/sec)	dP (kPa)	Efficiency (percent)
0	83	80
0.25	69	80
0.5	41	80

6. After the curve data is entered in the Raw Data table, select the Pressure Rise and Efficiency check boxes in the curve fitting section of the Compressor Configuration window (see Figure 3.5). This will cause curve fits for both pressure and efficiency to be calculated. Then, select the Generate Curve Fit Now button to generate the Compressor Performance graph (see Figure 3.6). Select the OK button on the Compressor Configuration window to accept the Compressor curve fit and to close the window. Select the OK button to close the Compressor/Fan Properties window.

J3, J4, J5, and J6 Branch Junctions

1. Open the Branch Properties window and enter an elevation of 0 meters for all of the Branch junctions.

J7 (Tool 1)

1. Enter the Name as Tool 1

2. Elevation = 0 meters

3. Set the "Valve Data Source" to "User Specified"

4. Set the "Subsonic Loss Model" to "Resistance Curve"

5. Check the optional Exit Valve Box

6. Exit Pressure = 0 barG

7. Exit Temperature = 21 deg. C

8. Select the Enter Curve Data button. Select the Flow Parameter as Mass flow in kg/sec. Enter 0.1 kg/sec and 55 kPa in the proper columns in the Raw Data table, then select the Fill As Quadratic button to generate the Valve loss curve data. Then select the Generate Curve Fit Now button to generate the valve loss curve. Select the OK button to accept the curve fit and to close the Valve Loss Curve Fit window and Valve Properties window.
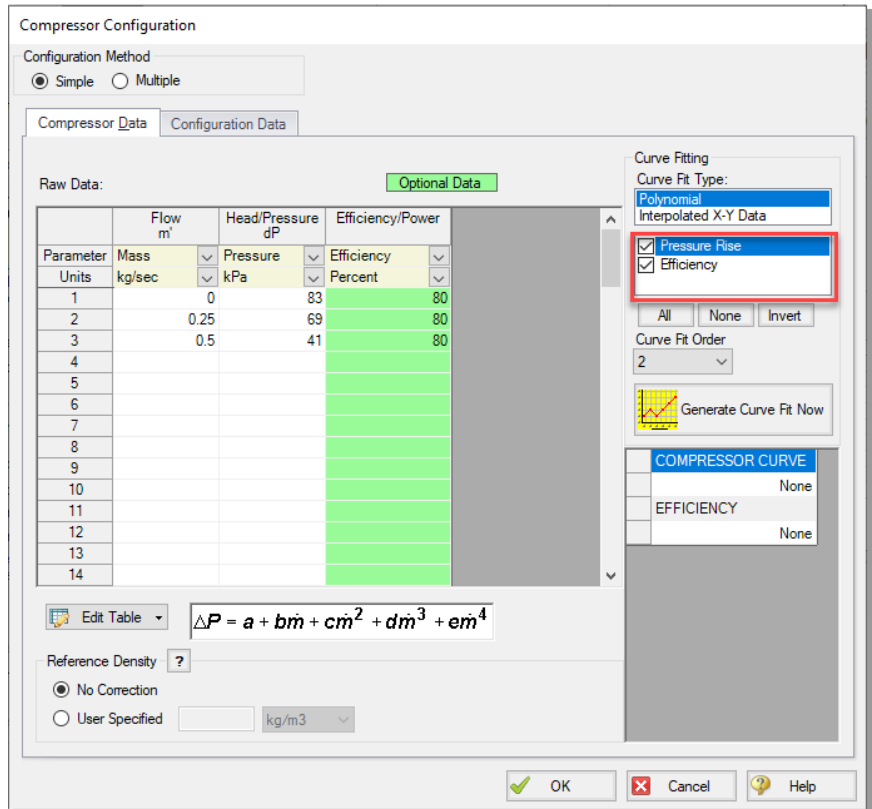

Figure 3.5   The Compressor Configuration window is where performance data is entered and fitted with a curve

J8, J9, J10 (Tool 2, Tool 3, and Tool 4)

1. For J8, open the Valve Properties window, and select J7 (Tool 1) from the "Copy Data From Jct…" list. This will open the Copy Data From Junction window. Click the "All" button. This will

cause all of the properties for J8 to be set the same as the properties for J7. Since you want the J8 tool to retain its unique name, click on the check box next to the Name parameter to deselect it (see Figure 3.7). Select the OK button. J8 now has all of the same properties as J7, with the exception of its name.

2. Enter the name as Tool 2

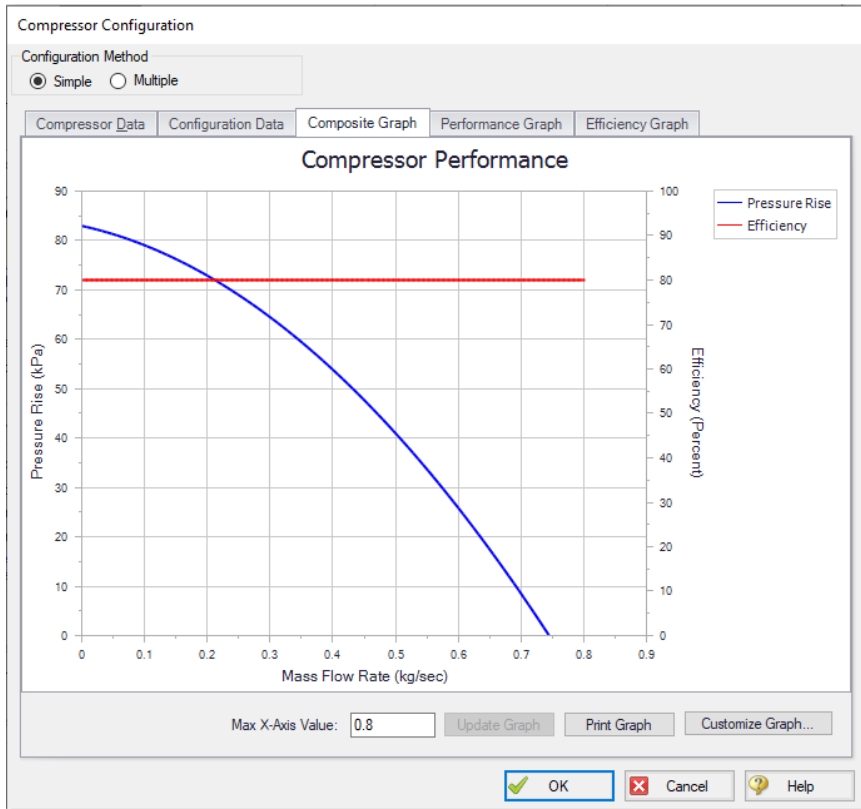3. Repeat this process for J9 and J10, entering names of Tool 3 and 4, respectively.

Figure 3.6   **Compressor Configuration window Composite Graph tab showing plots of curve fits generated from the Raw Data table**
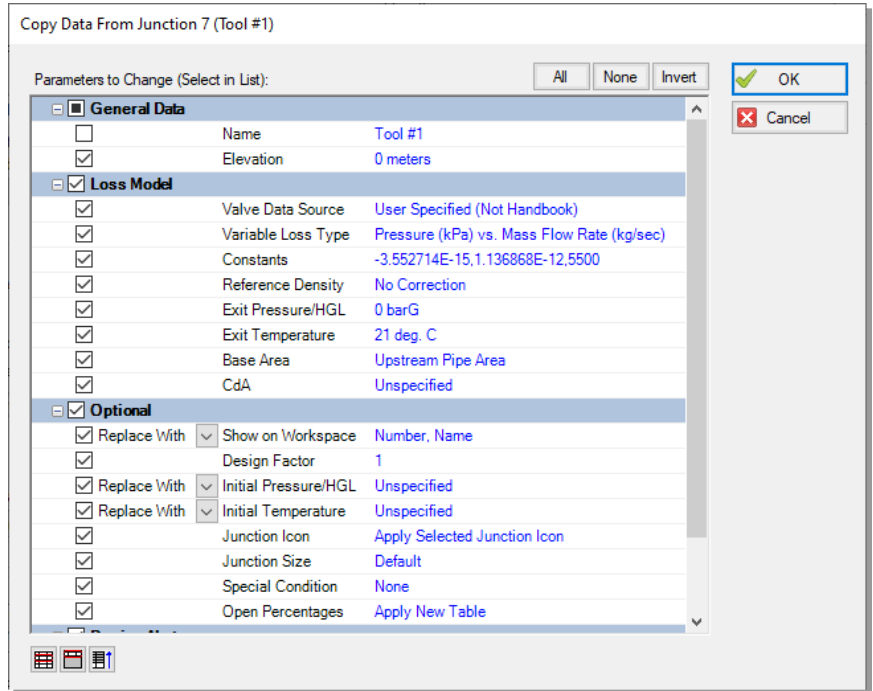
Figure 3.7 **The Copy Data From Junction window for junction J8 allows you to copy selected data from one junction to another**

D. Check if the pipe and junction data is complete

Check the Model Status light in the bottom right of the Quick Access Panel. A green light at this point indicates a fully defined model. A red light indicates undefined pipes or junctions in the model. If this happens, go back to the incomplete pipes or junctions and enter the missing parameters. Alternatively, open the List Undefined Objects window from the View menu to see the undefined objects.

Step 4. Create scenarios for the extreme temperature cases

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, 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

You will create two scenarios to model these cases. Scenarios are created using the Scenario Manager from the Quick Access Panel, or on the Tools menu. The Quick Access Panel, located on the right side of the Workspace area, contains the Scenario Manager on the Scenario tab. A scenario tree displays all model scenarios.

Click the "Create Child" icon on the Scenario Manager on the Quick Access Panel (see Figure 3.8). Name the child "Hot Case". A new scenario will appear below the Base Scenario in the scenario tree. Select the Base Scenario, create another child, and call it "Cool Case" (see Figure 3.8). Note that the new child scenario is loaded when it is created using the Quick Access Panel.

Step 5. Set up Hot Case scenario

Child scenarios "inherit" data from their ancestors. As long as the data has not been modified in a child scenario, data parameters in the child scenario will have the same value as their parent. Since the Base Scenario already contains all of the basic information for the model, such as the pipe and junction layout, pipe lengths, etc., you only need to modify the system conditions to simulate the case that will result in the hottest temperature at the tools.

Double-click the "Hot Case" scenario in the scenario tree of the Scenario Manager on the Quick Access Panel. This will load the "Hot

Case" scenario onto the Workspace as the current scenario. The currently loaded scenario is displayed in the scenario tree with a green check mark (Figure 3.8). Alternatively, you can use the Scenario Manager window from the Tools menu to load the scenario.
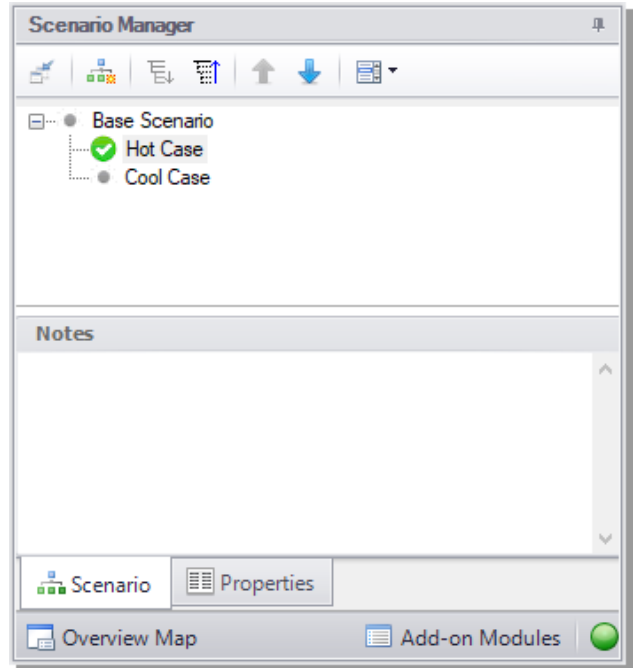

Figure 3.8   **The Scenario Manager on the Quick Access Panel displays the scenario tree that allows you to create model variants (scenarios) and keep them organized within the same model file**

In order to simulate the highest temperature case at the tools, you need to decide which design requirements will result in the highest tool temperatures. This will occur for the highest inlet and ambient temperatures for the air, and the heat transfer properties of the piping which will result in the least amount of heat loss.

Check the following model information as indicated:

J1 Atmosphere (Tank)

1.   Set the tank temperature is at the highest value of 45 deg. C.

J2 Compressor

1. The compressor will generate more heat if it is less efficient. Since this is the high temperature case, you should assume the worst possible efficiency, which is 80%. This data is entered on the Compressor/Fan Configuration window. Since you used 80% in the table when you added the Compressor information in the Base Scenario (Step 3), the compressor in this scenario has already inherited the Compressor curve for 80% efficiency.

Pipes P2 - P9

1. The heat transfer properties of the uninsulated pipes (P2 - P9), should be set to the values which will minimize the heat transfer occurring in the pipe, and result in the highest temperature at the tools.

2. For this case, you should use the highest ambient temperature of 24 deg. C and minimum velocity of 1.5 km/hr which will yield the lowest heat loss. Since you used this same data when you added the pipe heat transfer data in the Base Scenario using the Global Pipe Edit feature (Step 3), the pipes in this scenario have already inherited this heat transfer data, so no change in the heat transfer data is necessary. To verify this, click on Pipe P2 to open the Pipe Properties window. Select the Heat Transfer tab. The data should appear the same as shown in Figure 3.3.

Step 6. Run the Hot Case scenario

Select "Run Model" in the Analysis menu. This will launch the Solution Progress window. This window allows you to watch as the AFT Arrow Solver converges on the answer. When the solution has converged, view the results by pressing Output at the bottom of the Solution Progress window.

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 3.9. The junction output table shows the highest inlet static temperature at the tools for the Hot Case is 93.53 deg. C.

Figure 3.9   Output window for Hot Case Scenario. The Output window displays the Compressor/Fan Summary tab whenever a compressor/fan is modeled.

When compressors/fans are included in a model, AFT Arrow gathers the related data together in one convenient location in the Output window. This information can be found on the Compressor/Fan Summary tab located in the General Output section of the Output window, as shown at the top of Figure 3.9.

From Figure 3.9, the Compressor/Fan Summary shows that, for this system, the compression requirement of the compressor is 0.70 bar, and the mass flow is 0.27 kg/sec.

The Compressor/Fan Summary can also be displayed in a Results Diagram, as shown in Figure 3.10, by selecting the button next to "Show" in the Results Diagram column, double-clicking anywhere in the row, or right-clicking anywhere in the row and selecting Results Diagram.

Figure 3.10   Compressor Results Diagram

The pipe heat transfer results from the Heat Transfer tab (see Figure 3.11) can also be displayed in a Results Diagram, as shown in Figure 3.12, by selecting the button next to "Show" in the Results Diagram column, double-clicking anywhere in the row, or right-clicking anywhere in the row and selecting Results Diagram.
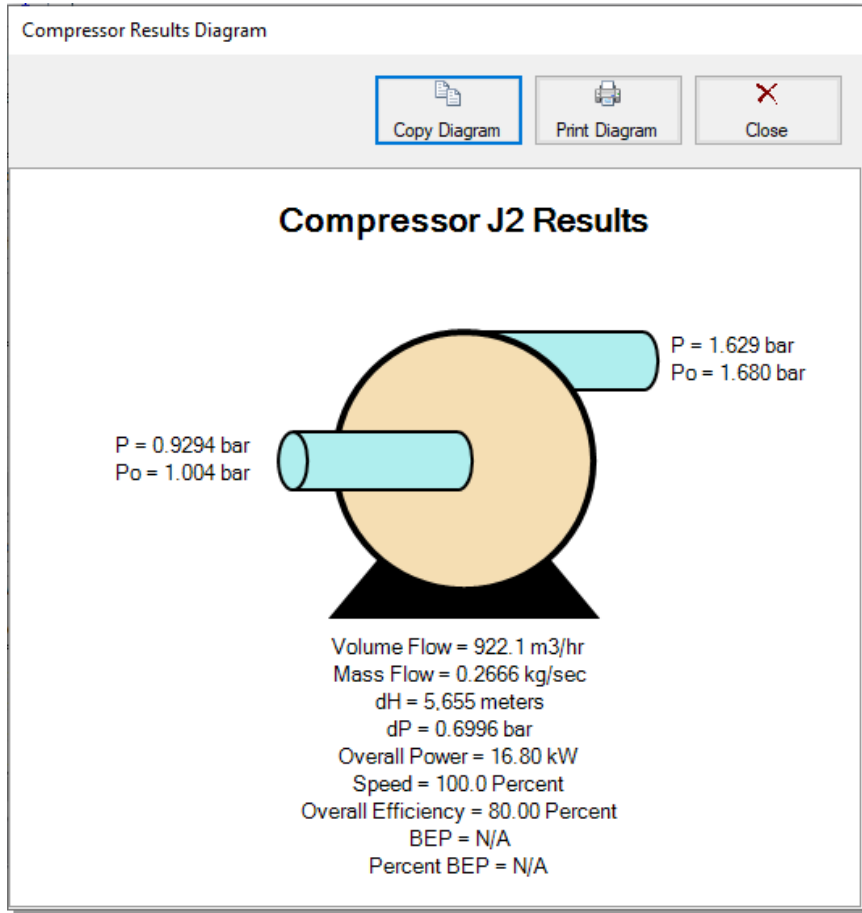
Pipe	Results Diagram		Name	T Fluid Inlet (deg. C)	T Fluid Outlet (deg. C)	T Inlet Wall Inside (deg. C)	T Outlet Wall Inside (deg. C)	T Inlet Outer Wall (deg. C)	T Outlet Wall Outside (deg. C)	T Ambient (deg. C)
1	Show	...	Pipe	38.17	38.03	N/A	N/A	N/A	N/A	N/A
2	Show	...	Pipe	104.18	99.54	105.22	101.40	105.16	101.34	24.00
3	Show	...	Pipe	101.53	96.23	100.65	95.75	100.59	95.70	24.00
4	Show	...	Pipe	97.55	90.50	94.50	87.84	94.45	87.80	24.00
5	Show	...	Pipe	91.24	79.54	84.96	74.36	84.92	74.33	24.00
6	Show	...	Pipe	98.80	93.53	101.12	97.51	101.05	97.44	24.00
7	Show	...	Pipe	94.29	89.28	95.96	92.22	95.89	92.16	24.00
8	Show	...	Pipe	87.63	82.99	89.05	85.49	88.99	85.44	24.00
9	Show	...	Pipe	76.17	72.24	77.74	74.76	77.69	74.71	24.00

Figure 3.11   The Heat Transfer tab in the Output window shows detailed heat transfer results for pipes
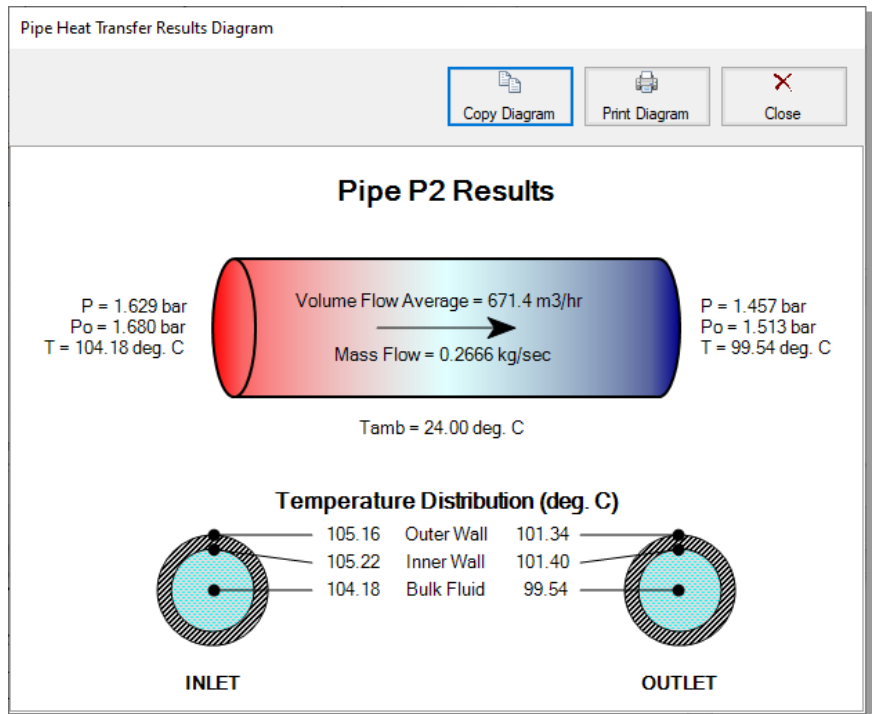

Figure 3.12   Pipe Heat Transfer Results Diagram

Step 7. Graph the Results

AFT Arrow allows you to create dual Y-axis graphs, which plot two parameters on the same graph.

In this example, you will graph the static temperature and the static pressure from the J1 tank through the pipeline to Tool 4 (J10) on a dual Y-axis graph to help you understand how the temperature and pressure change as the air flows through the pipeline.

Note: You can repeat this process for the flow path from the J1 tank to the other tools, or you can graph the flow through each path on this same graph using the Plot Multiple Paths Using Graphs feature.

1. From the Graph Control tab, choose the Profile tab in the Parameters/Formatting area, specify "Plot Single Path" in the Pipes definition area, then select pipes 1, 2, 3, 4, 5, and 9. These pipes make up the path from the J1 tank junction to Tool 4.

2. Ensure that meters is selected for the Length Units.

You will need to decide what parameter you wish to plot on the primary Y-axis and which parameter you want to plot on the secondary Y-axis. In this example, we will plot static pressure on the primary Y-axis and static temperature on the secondary Y-axis.

3. To do this, select "Pressure Static" in the Parameters definition area and choose "bar" as the static pressure unit.

4. Add a parameter by clicking on the "Add" button next to "Select Parameter" in the Parameters definition area.

5. Change this added parameter to "Temperature Static" and choose "deg. C" as the unit.

6. To specify that the temperature will be plotted on the secondary Y-axis, first select this parameter by clicking on the box to the left of "Temperature Static", then click on the "Make Secondary Y-Axis" icon (this is the blue, right arrow icon located at the top right over the selected parameters). Alternatively, you can double-click on the box to the left of "Temperature Static" in the Parameters definition area. Note that "Temperature Static" becomes indented under "Pressure Static", and that the black arrow next to "Temperature

Static" turns to point towards the graph. See Figure 3.13 to see how these graphing parameters are defined.
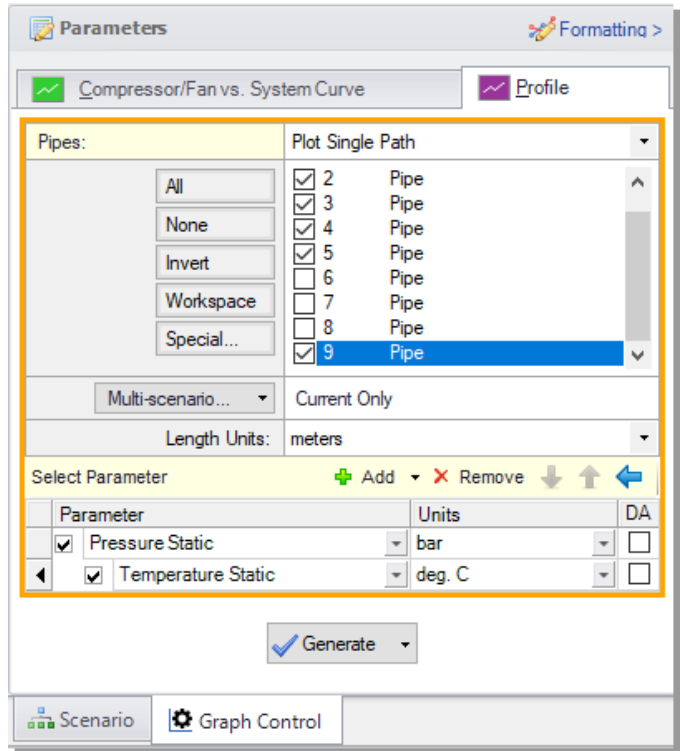
Figure 3.13   Parameter definition for dual Y-axes graph of static pressure (Primary Y-axis) and static temperature (Secondary Y-axis) on the Graph Control tab on the Quick Access Panel

Click the Generate button. You will see that the static pressure and temperature increase after the air flows through the compressor, and then these parameters gradually drop as the air flows through the pipes and to each tool. See Figure 3.14 for this graph.
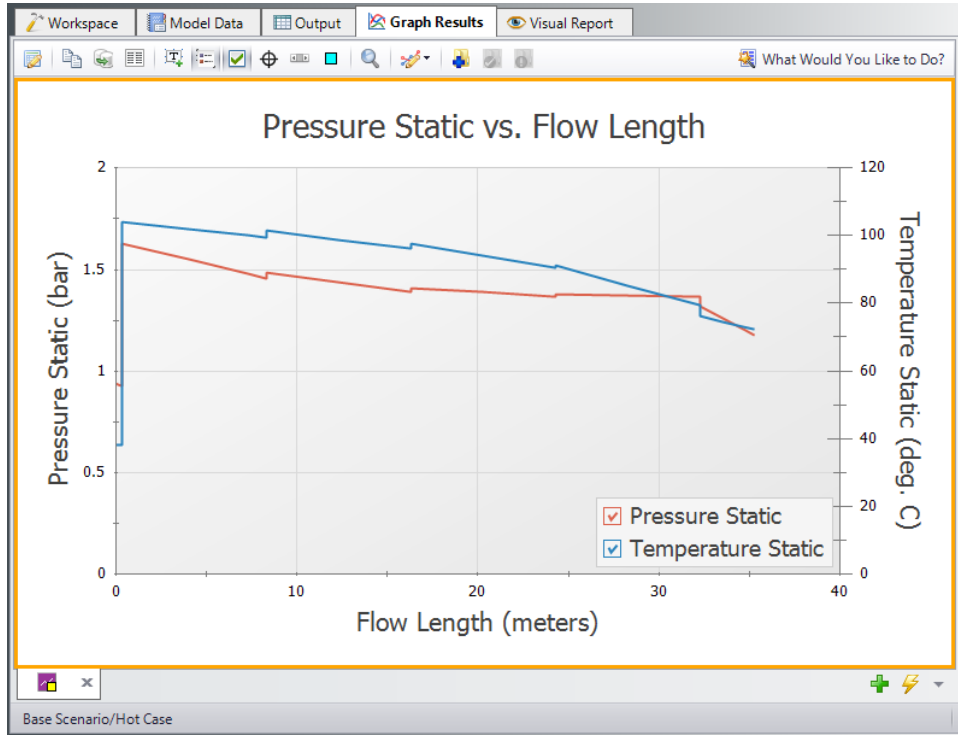
Figure 3.14   Pressure and temperature from the J1 tank junction to Tool #4 plotted on a dual Y-axis graph

Step 8. Set up Cool Case scenario

Double-click the "Cool Case" scenario in the list in the Scenario Manager on the Quick Access Panel. This will load the "Cool Case" scenario onto the Workspace as the current scenario.

In order to simulate the coolest temperature case, you must assume the lowest inlet and ambient temperatures, and the heat transfer properties for the piping that result in the largest amount of heat transfer.

Modify the following model information as indicated:

J1 Atmosphere (Tank)

1.   Set the temperature to lowest value of -18 deg. C.

J2 Compressor

1. The compressor will generate less heat if it is more efficient. Since this is the lower temperature case, you must assume the best possible efficiency, which is 90%. This data is entered on the Compressor/Fan Configuration window. When you added the compressor information in the Base Scenario (Step 4), the efficiency was entered in the table as 80%. Change the efficiency in the table to 90%. Select Efficiency under Curve Fitting, and then click the Generate Curve Fit Now button to generate a new efficiency curve. Close the Compressor Configuration and Properties windows by selecting the OK button.

Pipes P2 - P9

1. Set the heat transfer properties of the uninsulated pipes downstream of the compressor (P2 - P9) to the values that will result in the lowest gas temperatures at the tools. The appropriate ambient temperature to use is the lowest of 21 deg. C. 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 21 deg. C, we should use the highest ambient air velocity of 25 km/hr. This will permit maximum cooling of the gas in the pipe. On the other hand, if the compressor discharge temperature is less than 21 deg. C, then the lowest ambient air velocity of 1.5 km/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 25 km/hr. As just discussed, in practice one would need to verify the validity of this assumption.

2. Use the Global Pipe Edit feature to change the heat transfer properties in Pipes P2 - P9. Set the Ambient Temperature to 21 deg. C, and set the Fluid Velocity to 25 km/hr. Also, ensure that the Gnielinski correlation is selected under the Convection Coefficient column in the Fluid Internal row, and that Forced (Churchill-Bernstein) is selected in the External row.

Step 9. Run the Cool Case scenario

Select "Run Model" in the Analysis menu. When the solution has converged, view the results by clicking on the Output button at the bottom of the Solution Progress window.

The output for the cool Case Scenario is shown in Figure 3.15. The junction output table shows the lowest inlet static temperature at the tools for the cool Case is 18.5 deg. C. Note that the compressor discharge stagnation temperature is 26.1 deg. C (not shown in Figure 3.15), and thus the assumption of 25 km/hr for the air velocity is appropriate.

Figure 3.15   Output window for Cool Case Scenario

Analysis summary

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 a compressed air system. The temperature extremes of 18.5 deg. C to 93.53 deg. C can now be sent to the tool manufacturer, who will use it to compensate for the temperature sensitivity of the tools.

Isometric Pipe Drawing Mode

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. Figure 3.16 shows the compressed air system with the 2D grid displayed (select Show Grid from the Arrange menu).
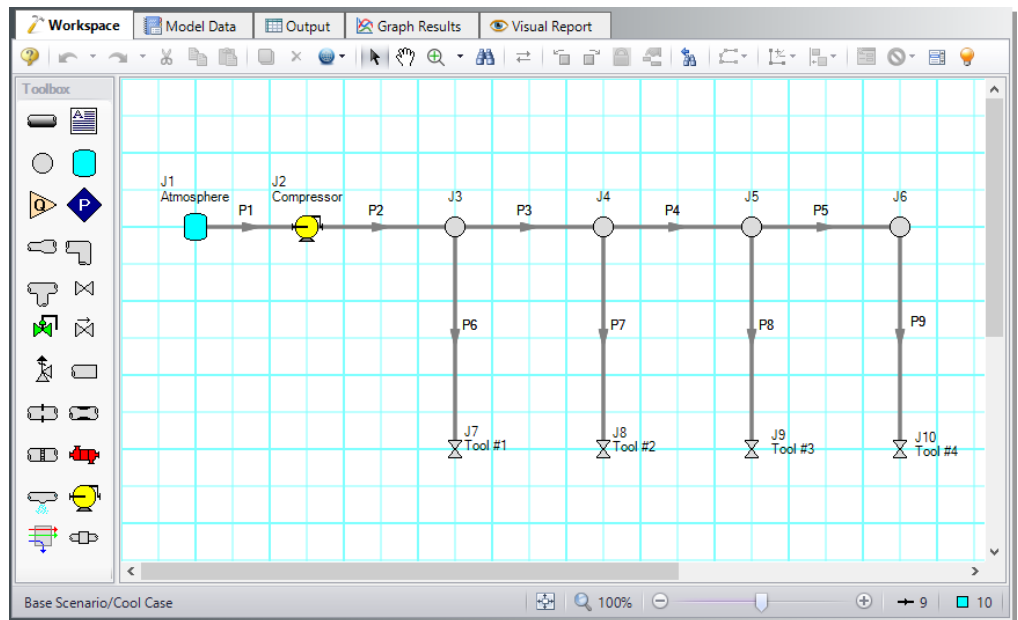
Figure 3.16   Compressed air system with 2D grid shown

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 3.17 shows the compressed air system built on an isometric grid.
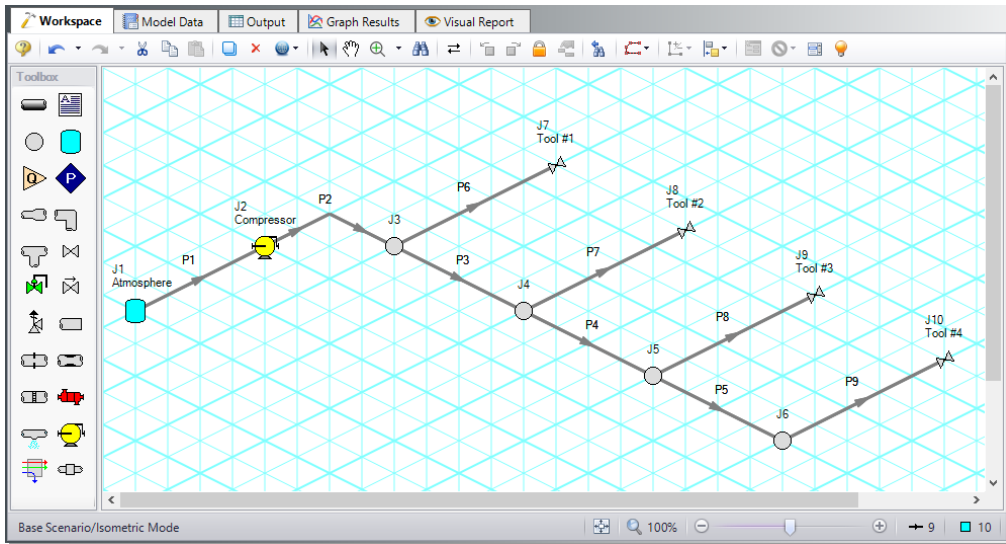
Figure 3.17   Compressed air system with isometric grid shown

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 3.17.

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. Select the icon shown in Figure 3.18.
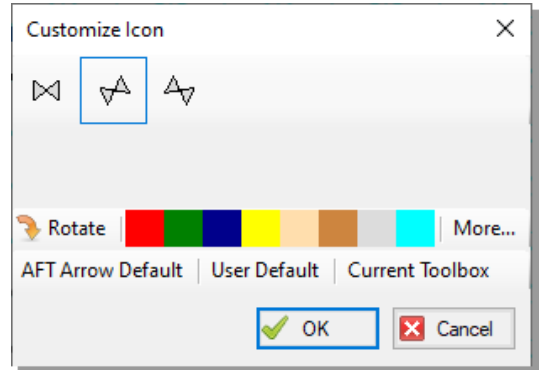
Figure 3.18   Right click on the junction to open the Customize Icon window and select the preferred settings

5.   Copy J7 for the tools, J8-J10, to maintain the preferred icon.

6.   Draw P1, as shown in Figure 3.19.

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 3.19 shows Pipe 2 being drawn with the preview line.
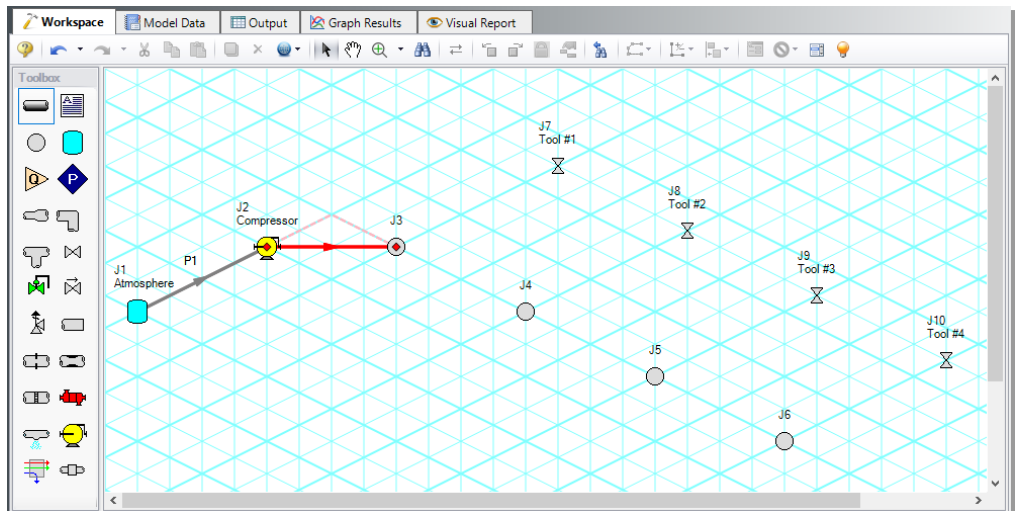
Figure 3.19   A preview line shows when drawing or adjusting pipes

Note: You can hold the "Alt" key while adjusting a pipe by the endpoint to add an additional segment. This can be used with the arrow key or mouse scroll wheel to change between different preview line options.

8.   Draw pipes P3-9, as shown in Figure 3.20.

9.   The grid can be shown or turned off in the Arrange menu.
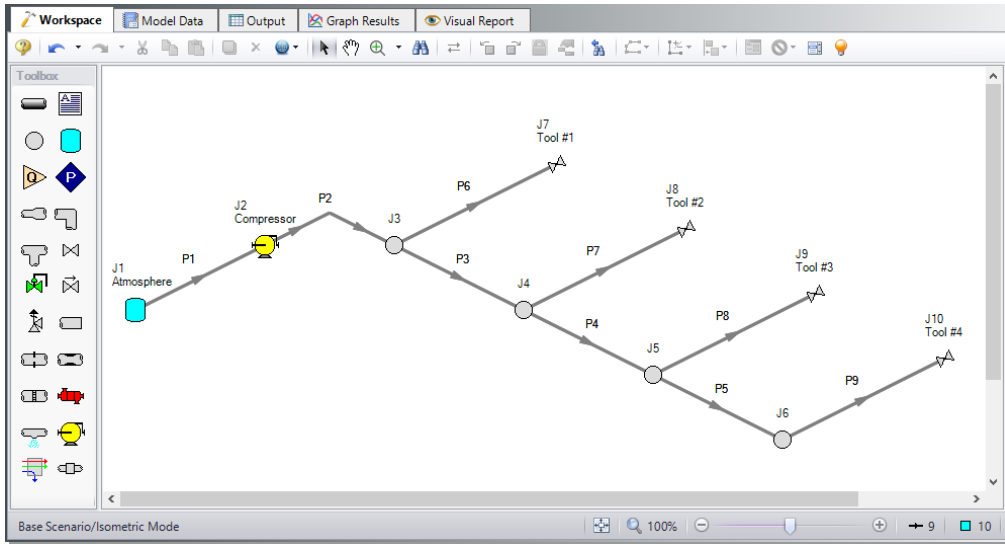
Figure 3.20   Compressed air system with isometric grid not shown

Natural Gas Burner System Example

This example demonstrates how to size a heat exchanger to deliver natural gas at a minimum required temperature.

Topics covered

This example will cover the following topics:

- Sizing a heat exchanger for a particular system

- Entering a loss curve for a heat exchanger

- Specifying heat transfer data for pipes

- Use of flow control valves

Required knowledge

This example assumes that the user has some familiarity with AFT Arrow such as placing junctions, connecting pipes, and entering pipe and junction properties. Refer to the Flow Through an Orifice Example in Chapter 2 for more information on these topics.

Model file

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

- *Metric – Natural Gas Burner.aro* - AFT Arrow model file

This example also provided in metric units in the *US – Natural Gas Burner.aro* model file.

Problem statement

An underground storage reservoir containing natural gas made up mostly of methane supplies gas to five burners. The gas source is at 34.5 barG (3450 kPa(g)), and 10 deg. C. The gas is supplied to each of the burners at 6.9 barG (690 kPa(g)). To assure that the flow to each of the burners is equal, each burner has a control valve that controls the flow to 2.25 kg/sec.

The pipes in the system are uninsulated STD (schedule 40) steel pipes, with standard roughness. The ambient temperature surrounding the pipes is 24 deg. C, with an air velocity of 25 km/hr.

In order to operate at the best efficiency, the gas should be delivered to the burners at a minimum stagnation temperature of 38 deg. C. To achieve this, the gas flows through a heat exchanger which has a 70 kPa pressure drop at 11 kg/sec.

How much heat must be added by the heat exchanger to ensure that all of the burners will receive gas at the specified minimum temperature? Elevation changes can be neglected.

Step 1. Start AFT Arrow

From the Start Menu choose AFT Products and AFT Arrow.

Step 2. Specify fluid properties

1. Open Analysis Setup by selecting Analysis Setup from the common toolbar or the Analysis menu.

2. Under the Fluid Properties group open the Fluid panel if not already open. Select the AFT Standard library and then select "Methane" in the Fluids Available in Library list.

3. Click "Add to Model" to select methane for use in this model

4. Use the default "Redlich-Kwong" for the equation of state and "Generalized" for the enthalpy model

5. Click OK

Step 3. Build the model

A. Place the pipes and junctions

This completes the first two groups of the Analysis Setup. The Pipes and Junctions group remains undefined. To complete this group, begin by assembling the model on the Workspace as shown in Figure 4.1.
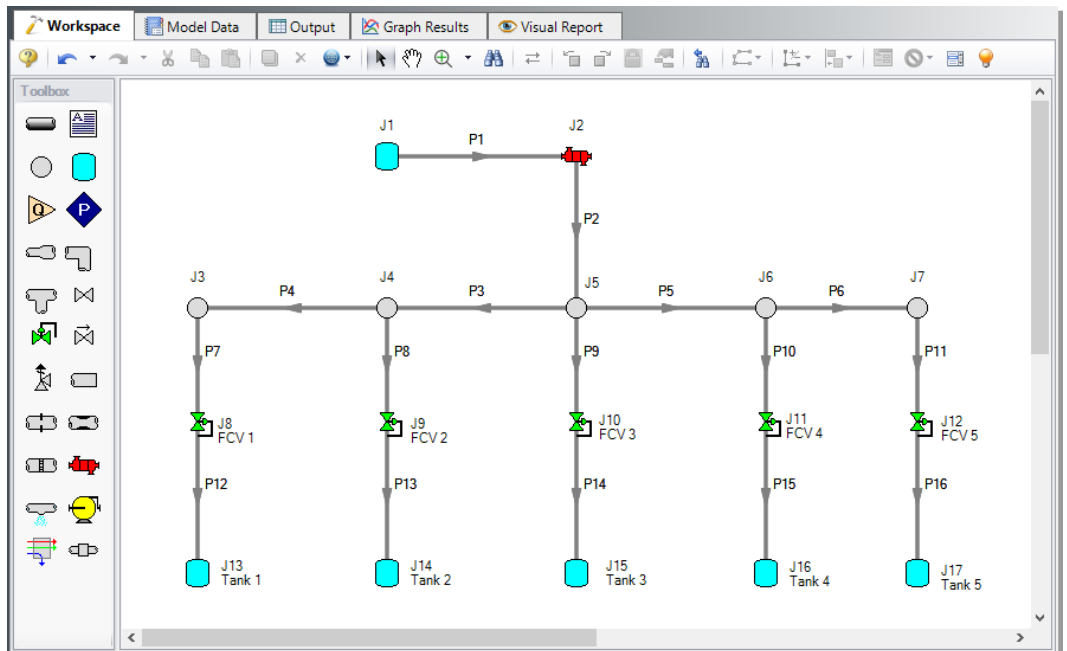
Figure 4.1   Layout of pipe system for Natural Gas Burner System Example

B. Enter the pipe data

The system is in place, but now you need to enter the input data for the pipes and junctions. Double-click each pipe and enter the following data

in the Properties window (or use the Global Pipe Editing window, as discussed in Chapter 3).

All of the pipes are Steel – ANSI, STD (schedule 40), with the Standard friction model and the following data:

Pipe	Length (meters)	Nominal Size
P1	30	4 inch
P2 - P6	6	4 inch
P7 - P16	1.5	3 inch

Enter the following data on the Heat Transfer tab for each pipe, as shown for P1 in Figure 4.2:

1. Heat Transfer Model = Convective Heat Transfer

2. Ambient Temperature = 24 deg. C

3. Fluid Velocity = 25 km/hr

4. Fluid = Air
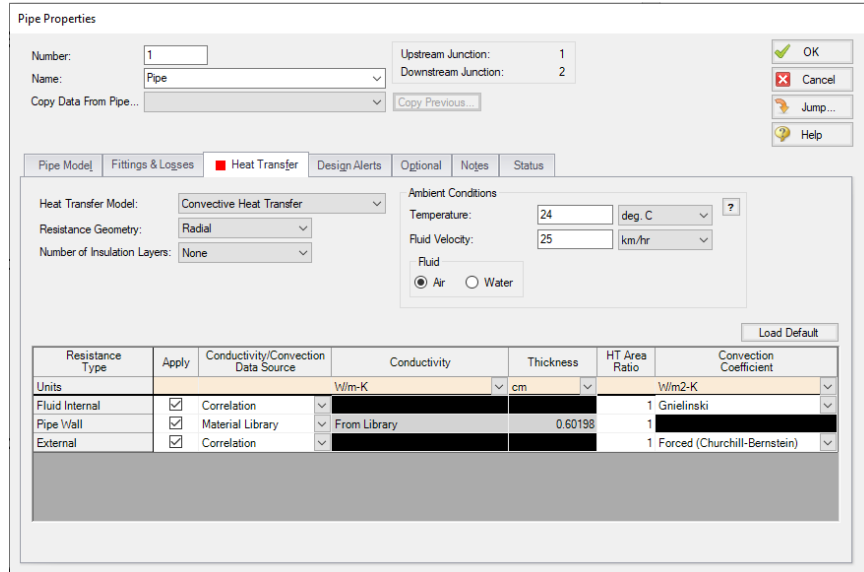
Figure 4.2   Pipe 1 Heat Transfer window

C. Enter the junction data

J1 Supply Reservoir (modeled as a Tank junction)

1. Elevation = 0 meters

2. Pressure = 34.5 barG (3450 kPa(g))

3. Temperature = 10 deg. C

J2 Heat Exchanger

1. Elevation = 0 meters

2. Enter Loss Model Data

   a. Select the Loss Model as "Resistance Curve", and then click the "Enter Curve Data" button

   b. Select the "Flow Parameter" as "Mass"

   c. Enter 11 kg/sec and 70 kPa in the Raw Data table

   d. Click the "Fill as Quadratic" button to create data for a square loss curve

   e. Select "Generate Curve Fit Now" to create a quadratic curve fit

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 Thermal Data tab on the Heat Exchanger Properties window, and select the Thermal Model as "Specified Heat Rate In Constant". Then enter a value for Heat Flow Into System. For the first guess, use 500 kW.

J8 - J12 flow control valves

1. Elevation = 0 meters

2. Valve Type is Flow Control (FCV)

3. Control Setpoint, Mass Flow Rate = 2.25 kg/sec

J13 - J17 burners (modeled as Tank junctions)

1. Elevation = 0 meters

2. Pressure = 6.9 barG (690 kPa(g))

3. Temperature = 260 deg. C

J3 - J7 branch junctions

1. Elevation = 0 meters

D. Check if the pipe and junction data is complete

Check the Model Status light in the bottom right of the Quick Access Panel. A green light at this point indicates a fully defined model. A red light indicates undefined pipes or junctions in the model. If this happens, go back to the incomplete pipes or junctions and enter the missing parameters. Alternatively, open the List Undefined Objects window from the View menu to see the undefined objects.

Step 4. Run the model

Select "Run Model" in 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 model has converged, you can view the results by clicking on the Output button at the bottom of the Solution Progress window.

Step 5. Examine the results

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

Examination of the stagnation temperature at the outlets of the pipes discharging to the burners shows that the lowest temperature is 16.93 deg. C (see Pipe 16 in Figure 4.3). This is well below the required minimum temperature of 38 deg. C. The heat added to the heat exchanger must be increased, and the model must be run again to determine the outlet temperature at the new value.

Step 6. Iterate on heat rate

By trial and error, you will find that you need to add 1052 kW to the heat exchanger to achieve a minimum stagnation temperature of 38 deg. C.

Analysis summary

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

The goal seeking capabilities in the optional 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.

More discussion on the GSC Module is given in Chapter 5.

Figure 4.3   Output for Natural Gas Burner System Example

AFT Arrow Add-on Modules Examples

This chapter covers two examples, one that demonstrates how to use the Goal Seek and Control (GSC) add-on module, and one that demonstrates how to use the Extended Time Simulation (XTS) add-on module.

The user can only perform these examples if access to the relevant modules are available.

Topics covered

This example will cover the following topics:

- Defining GSC Variables and Goals

- Using Group Max/Min Goals

- Linking Variables

- Specifying the Transient Control group

- Understanding Transient Output

Required knowledge

This example assumes that the user has some familiarity with AFT Arrow such as placing junctions, connecting pipes, and entering pipe and junction properties. Refer to the Flow Through an Orifice example in Chapter 2 for more information on these topics. To model the heat transfer in these examples, the user may need to refer to Chapter 3 for more detailed explanations.

Model files

These examples use the following files, which are installed in the Examples folder as part of the AFT Arrow installation:

- *Metric - Process Steam System - GSC.aro* - AFT Arrow model file

- *Metric - Air Receiver Tank - XTS.aro* - AFT Arrow model file

GSC Problem statement

A process steam supply system has two boilers at 20.7 barG (2070 kPa (g)) and 430 deg. C.

All of the pipes in the system have 5 cm of insulation with a thermal conductivity of 0.03 W/m-K. The ambient temperature is 24 deg. C with an average air velocity of 11 km/hr. 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.

Assume "Redlich-Kwong" for the equation of state model, and "Generalized" for the enthalpy model. All of the tees in the system should be modeled using the Detailed loss model, with sharp edges.

The system has six users with variable peak flow demands.

- User #1: 18,000 kg/hr

- User #2: 23,000 kg/hr

- User #3: 25,000 kg/hr

- User #4: 25,000 kg/hr

- User #5: 18,000 kg/hr

- User #6: 27,000 kg/hr

For the specified design flows, the process steam system must meet a minimum delivery pressure of 7 barG (700 kPa(g)) at each discharge.

GSC Step 1. Start AFT Arrow

From the Start Menu choose AFT Products and AFT Arrow.

GSC Step 2. Specify solution control

There is an option available in the Solution Method panel when using one of the marching methods that can sometimes result in an overall reduction in solution run time. 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.

To activate this option, do the following:

1. Open the Analysis Setup window from the toolbar

2. Under the Steady Solution Control group on the Solution Method panel, select the "First Use Lumped Adiabatic Method to Obtain Initial Stating Point For Marching Solution" option

GSC Step 3. Specify fluid properties

1. In the Analysis Setup window on the Fluid panel, select the AFT Standard library, and then select "Steam" in the fluids available window

2. Click "Add to Model" to select steam for use in this model

3. Select the "Redlich-Kwong" Equation of State option

4. Select the "Generalized" Enthalpy model

5. Click the OK button

GSC Step 4. Build the model

A. Place the pipes and junctions

This completes the first two groups of the Analysis Setup. The Pipes and Junctions group remains undefined. To complete this group, begin by assembling the model on the Workspace as shown in Figure 5.1.
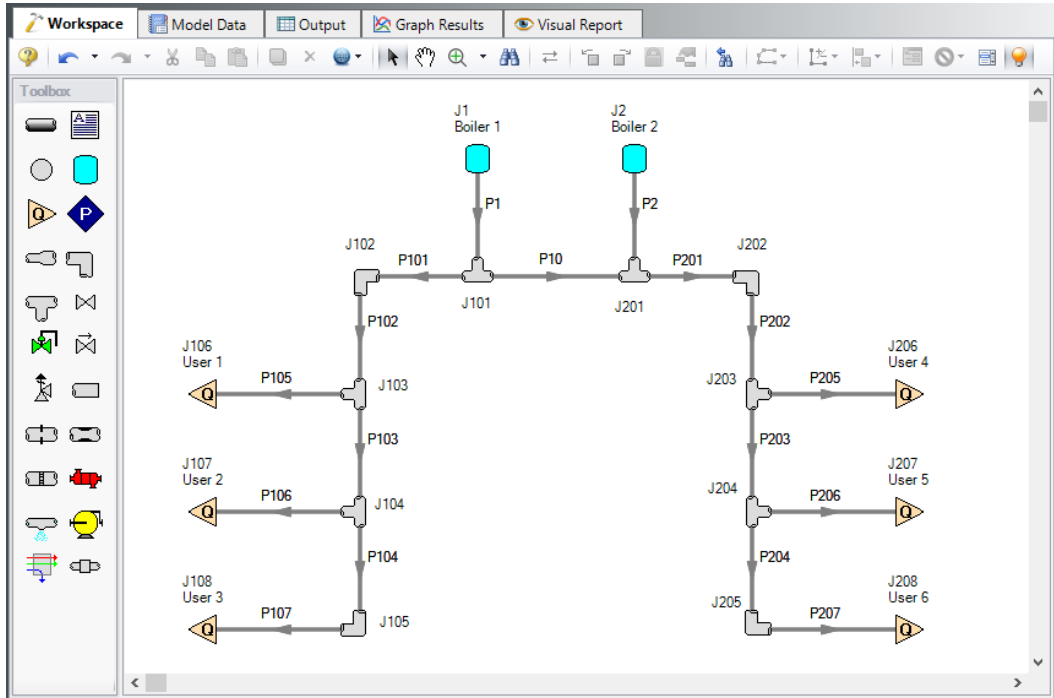
Figure 5.1   Layout of pipe system for Process Steam System Example

B. Enter the pipe data

The system is in place, but now you need to enter the input data for the pipes and junctions. Double-click each pipe and enter the following data in the Properties window (or use the Global Pipe Editing window, as discussed in Chapter 3):

All of the pipes are Steel – ANSI STD (Schedule 40) (Note: The 12 inch diameter pipes will be Schedule 40) with standard roughness:

Pipe	Length (meters)	Nominal Size
P1	30	12 inch
P2	46	12 inch
P10	23	12 inch
P101	122	8 inch
P102	229	8 inch
P103	229	8 inch
P104	76	8 inch
P105	30	6 inch
P106	15	6 inch
P107	23	6 inch
P201	76	8 inch
P202	152	8 inch
P203	152	8 inch
P204	152	8 inch
P205	15	6 inch
P206	23	6 inch
P207	15	6 inch

Enter the following data on the Heat Transfer tab for each pipe:

1. Heat Transfer Model = Convective Heat Transfer

2. Ambient Temperature = 24 deg. C

3. Fluid Velocity = 11 km/hr

4. Number of Insulation Layers = 1 External

5. Insulation #1 Conductivity = 0.03 W/m-K, Thickness = 5 cm

C. Enter the junction data

J1, J2 Tank Junctions

1. Elevation = 0 meters

2. Pressure = 20.7 barG (2070 kPa(g))

3. Temperature = 430 deg. C

J106, J107, J108, J206, J207, J208 Assigned Flow Junctions

1. Elevation = 0 meters

2. Flow Rate = see Problem Statement

J102, J105, J202, J205 Bend Junctions

1. Elevation = 0 meters

2. Type = Smooth Bend

3. Angle = 90 deg.

4. r/D = 1.5

Note: The arrangement of the pipes on the Detailed tee junctions is important. Be certain that the inlet, outlet and branch pipes are assigned appropriately.

J101, J103, J104, J201, J203, J204 Tee Junctions

1. Elevation = 0 meters

2. Loss Model = Detailed

3. Type = Tee (Sharp, Straight Run)

4. Initial Pressure = 7 barG (700 kPa(g)) (on Optional tab)

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

D. Check if the pipe and junction data is complete

Check the Model Status light in the bottom right of the Quick Access Panel. A green light at this point indicates a fully defined model. A red light indicates undefined pipes or junctions in the model. If this happens, go back to the incomplete pipes or junctions and enter the missing parameters. Alternatively, open the List Undefined Objects window from the View menu to see the undefined objects.

GSC Step 5. Create a scenario for GSC case

Create a new scenario from the Base Scenario for the GSC Case. Refer to Chapter 3 for directions on how to add a scenario.

GSC Step 6. Activate the GSC Module

Open the Analysis Setup window and go the the Modules panel under the Modules group. Activate the GSC module by selecting "Activate GSC" as is shown in Figure 5.2.

Though the module is activated, you have the option to use or ignore it. This is helpful if you want to check the impact of changes in your system configuration or in troubleshooting the model without solving for the variables. For the module's features to be used in the simulation calculations, set the GSC module to Use under the Activate GSC checkbox (Figure 5.2). The Use and Ignore options can also be toggled from the Analysis menu Goal Seek & Control item.
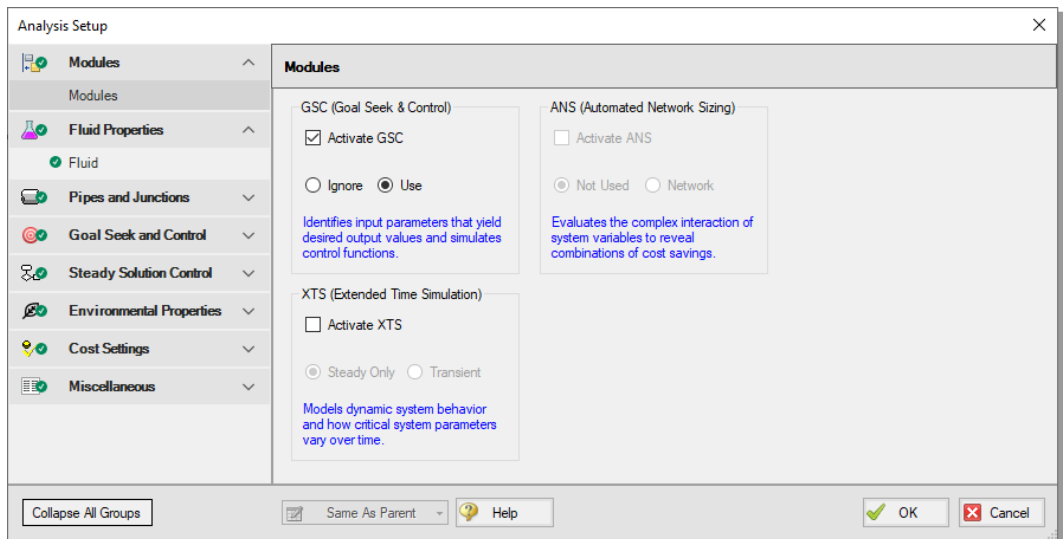

Figure 5.2   **Activating the GSC module and setting it to be used in AFT Arrow**

GSC Step 7. Goal Seek and Control group

When the GSC Module is activated, the Goal Seek and Control group appears in the Analysis Setup window with the Variables, Goals, and Numerical Controls panels.

The Goal Seek and Control settings are specified by entering data for variables and goals in the Goal Seek and Control group panels. Go to the Variables panel as shown in Figure 5.3 below.

GSC Step 8. Add the variables

In the GSC module, variables are the parameters that AFT Arrow will modify in order to achieve the specified goals. To obtain a unique answer, there should be one variable applied for each goal. You may define as many variables and goals as you wish.

The Variables panel allows users to create and modify the system variables. On the Variables panel, you are able to select the object and junction types, the name and number of the object to which the variable applies, and the object parameter that is to be varied. Additionally, you can elect to apply certain variables while omitting others.
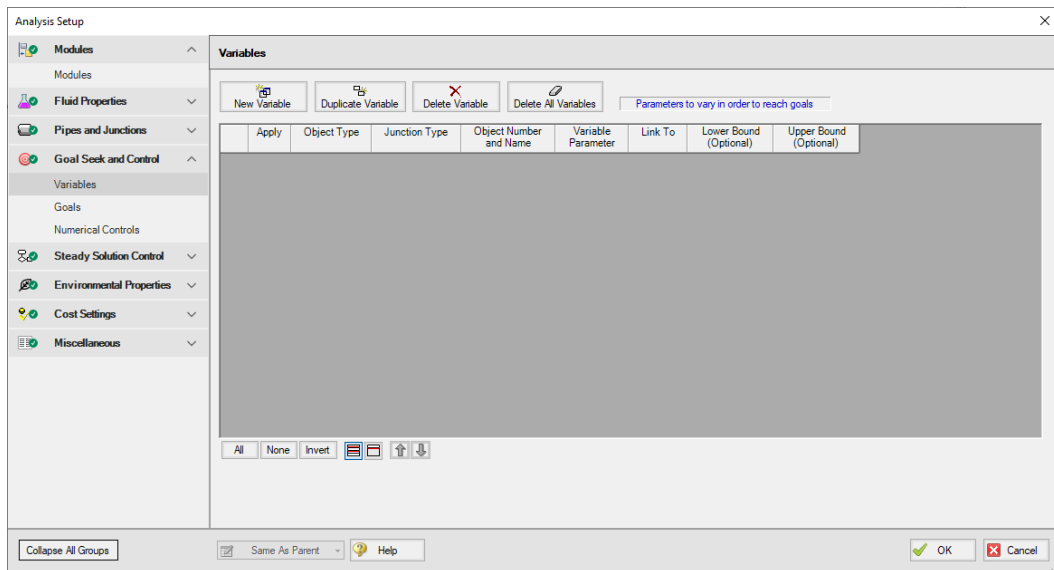

Figure 5.3   The Goal Seek and Control group is used to define GSC Variables and Goals

For this example, we want to find the supply pressures that will give us a specific minimum discharge pressure. Select the "New Variable" button, and input the following variable data:

1. Apply: Checked

2. Object Type: Junction

3. Junction Type: Tank

4. Junction Number and Name: J1 (Boiler 1)

5. Variable Parameter: Pressure

6. Link To: (None)

7. Lower and Upper Bounds: Leave blank

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.

Linking variables

This process steam system has two boilers to supply steam to the users. We have defined a variable for the first boiler, but we have not yet accounted for the second boiler.

For this example, the discharge pressures will be defined as a single Group Max/Min goal (see Step 5). However, if we define the second boiler as another independent variable, this would create 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.

Select the "New Variable" button to add the variable for the second boiler. To implement variable linking, specify Boiler 1 (J1) in the Link To column of the Boiler 2 (J2) variable. Enter the data as shown below.

1. Apply: Checked

2. Object Type: Junction

3. Junction Type: Tank

4. Junction Number and Name: J2 (Boiler 2)

5. Variable Parameter: Pressure

6. Link To: J1 (Boiler 1)

7. Lower and Upper Bounds: Leave Blank

After entering the data, the Variables panel should appear as shown in Figure 5.4. Note the variables panel will appear as undefined until at least one goal is applied.
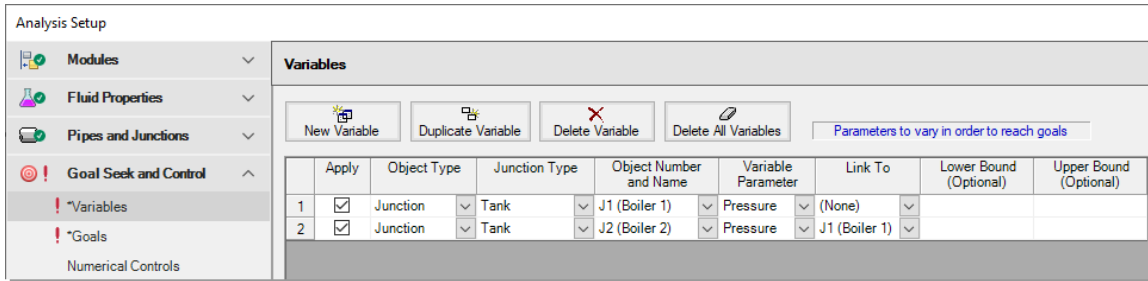
Figure 5.4   GSC Variables are parameters that are changed by AFT Arrow to achieve the defined goals

GSC Step 9. Add a goal

Goals are the parameters you want to achieve. The goals are achieved as AFT Arrow modifies the variables. To obtain a unique answer, there should be one goal applied for each variable. You may define as many variables and goals as you wish.

Select the Goals panel in the Goal Seek and Control group. 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 (e.g., the inlet or outlet of a pipe object).

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 7 barG (700 kPa(g)).

Define a group

In order to use a Group goal, the objects that the goal applies to must be added to a Group. For detailed information about Groups, see the AFT Arrow Help system.

To create a group, select the Groups > Create option from the Edit menu (note: you will have to click OK on the Analysis Setup window to be able to access the menu items). When prompted, give the new group a name, such as "Users", and select OK. Alternatively, you can select the Assigned Flow junctions on the Workspace before creating the groups and they will be pre-selected when the group is created (note: you can select multiple objects at the same time by holding SHIFT as you select each object).

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 5.5. After choosing the assigned flow junctions, select the OK button on the Group Manager window.
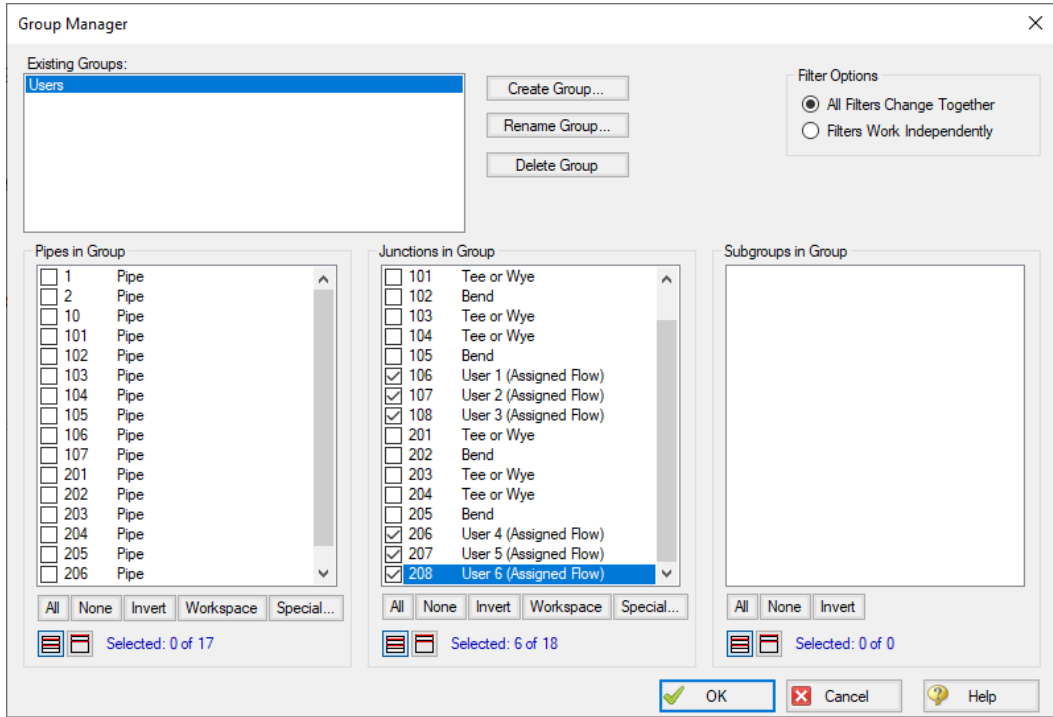
Figure 5.5   Groups are created using the Group Manager

Add a Group Max/Min goal

Now that the group has been defined, reopen the Analysis Setup window, and select the Goals panel. Add a new Goal by clicking the New Goal button, and entering 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: 7

7. Goal Units: barG

8. Object ID: Users

9. Object Location: NA

After entering the data, the Goals tab should appear as shown in Figure 5.6.

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. This is illustrated in Figure 5.7.

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.
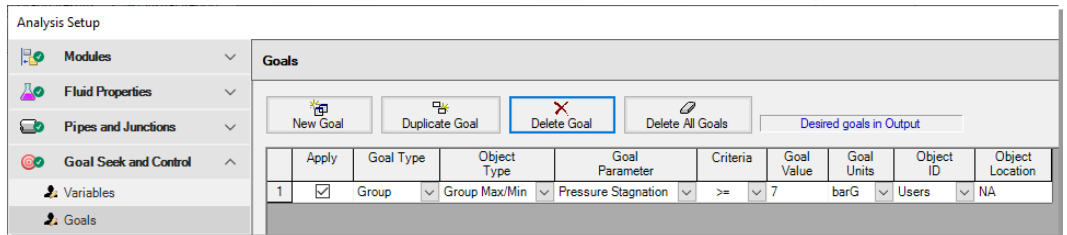

Figure 5.6   GSC Goals are the parameter values set by the user that AFT Arrow adjusts the variables to achieve

GSC Step 10. Run the model

Select "Run Model" in 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.

Note: When using the GSC module there is a new area displayed in Solution Progress that shows the specific progress of the GSC module. As it makes progress, the Best (Lowest) value will decrease towards zero. The field in the far right displays how many complete hydraulic solutions have been run.

After completion click the Output button at the bottom of the Solution Progress window.
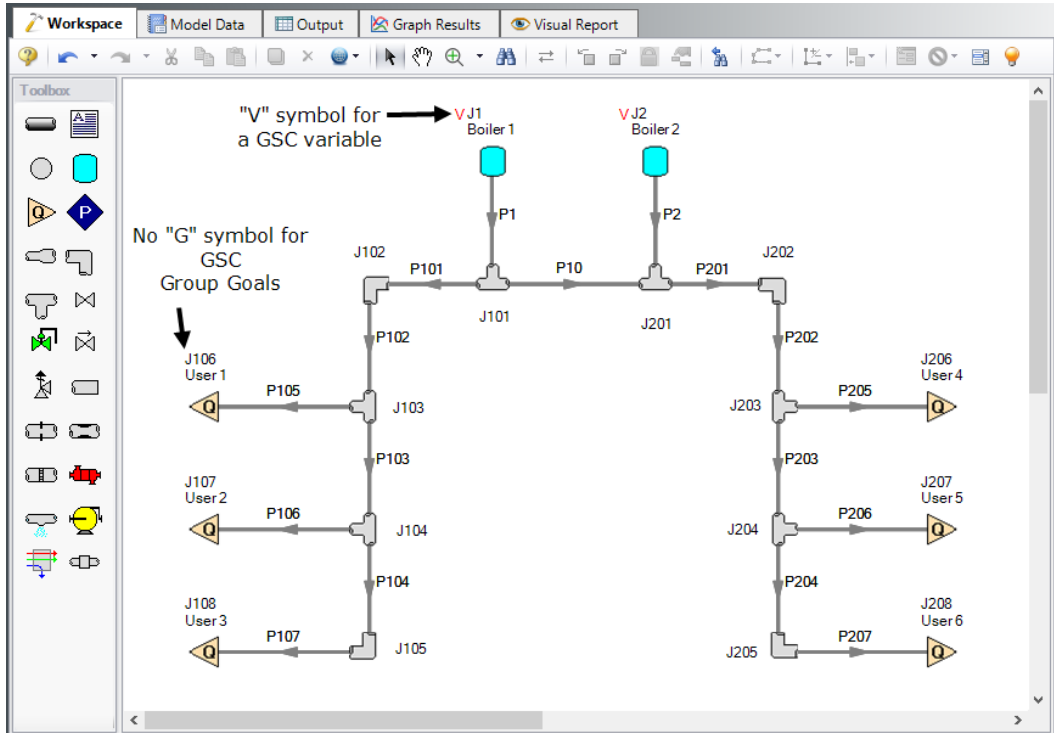
Figure 5.7   AFT Arrow displays symbols next to objects on the Workspace that have goals or variables defined

GSC Step 11. Examine the results
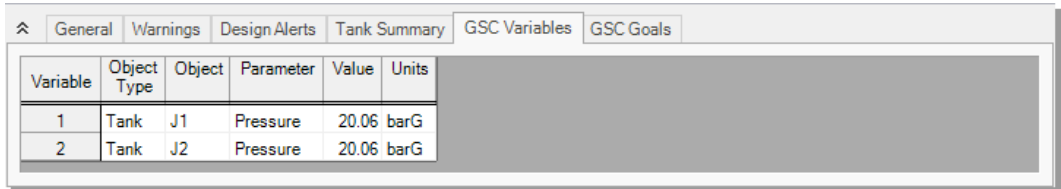
The Output window contains all the data that was specified in the Output Control window. The results of the GSC analysis are shown in the General Output section.

The GSC Variables tab shows the final values for the variable parameters, as shown in Figure 5.8. The GSC Goals tab shows the values achieved for the goals, as shown in 5. .

GSC analysis summary

The GSC module analysis determined the supply pressure required to deliver a minimum pressure of 7 barG at all of the users is 20.06 barG at

the boilers. Note that, because the variable for the J2 Boiler pressure was linked to the J1 Boiler variable, the final variable results for both are identical.

Figure 5.8   The final GSC Variable values are shown on the GSC Variables tab in the Output window General section

XTS Problem Statement

An initially unpressurized tank is being charged with compressed air for use in a compressed air system. The compressed air tank needs to be charged to a pressure of 8.5 barG.

An actively cooled positive displacement compressor rated for 170 sm3/h is used in the system with an aftercooler which is sized to cool the air to 30 deg. C. The cooler has a pressure drop of 20 kPa at 170 sm3/h. There is an isolation valve at the air receiver tank inlet to control flow out of the tank during normal operations with a Cv of 50 and Xt of 0.7.

Atmospheric air is assumed to be dry, with an ambient temperature of 21 deg. C. Determine the time it will take to charge the air receiver tank for this process.

XTS Step 1. Start AFT Arrow

From the Start Menu choose AFT Products and AFT Arrow.

XTS Step 2. Activate the XTS Module

Open the Analysis Setup window and go the the Modules panel under the Modules group. Activate the XTS module by selecting "Activate XTS" as is shown in Figure 5.9.

Though the module is activated, you have the option to set the analysis to Steady Only or Transient. This is helpful to check the steady state results for the model without running the transient simulation. To run the Transient analysis, set the XTS module to Transient under the Activate XTS checkbox (Figure 5.9). The Steady Only and Transient options can also be toggled from the Analysis menu Time Simulation item.

When the XTS module is activated a new group will appear in Analysis Setup titled Transient Control.
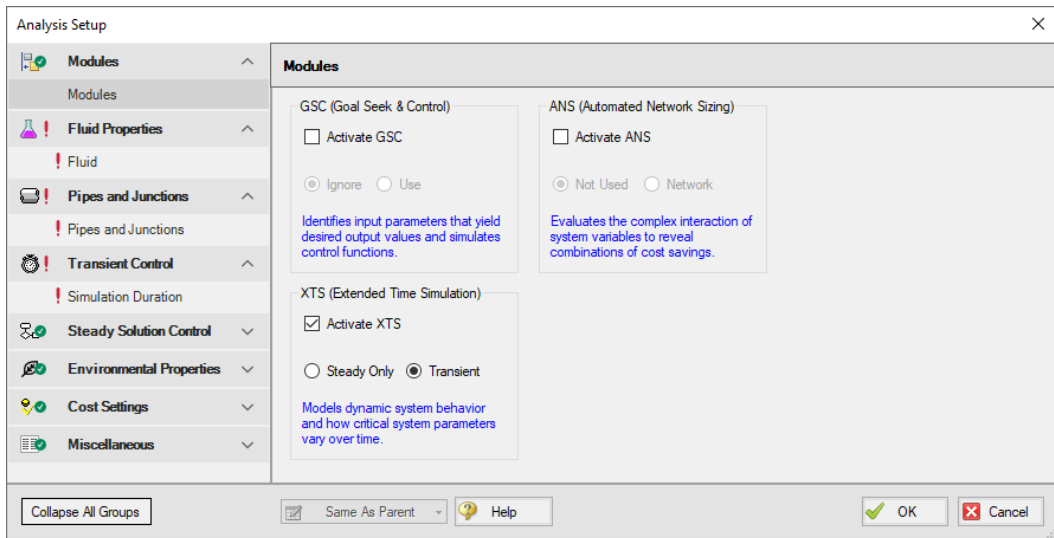

Figure 5.9   Activating the XTS module and setting the simulation type to Transient

XTS Step 3. Define simulation duration

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 5.10.
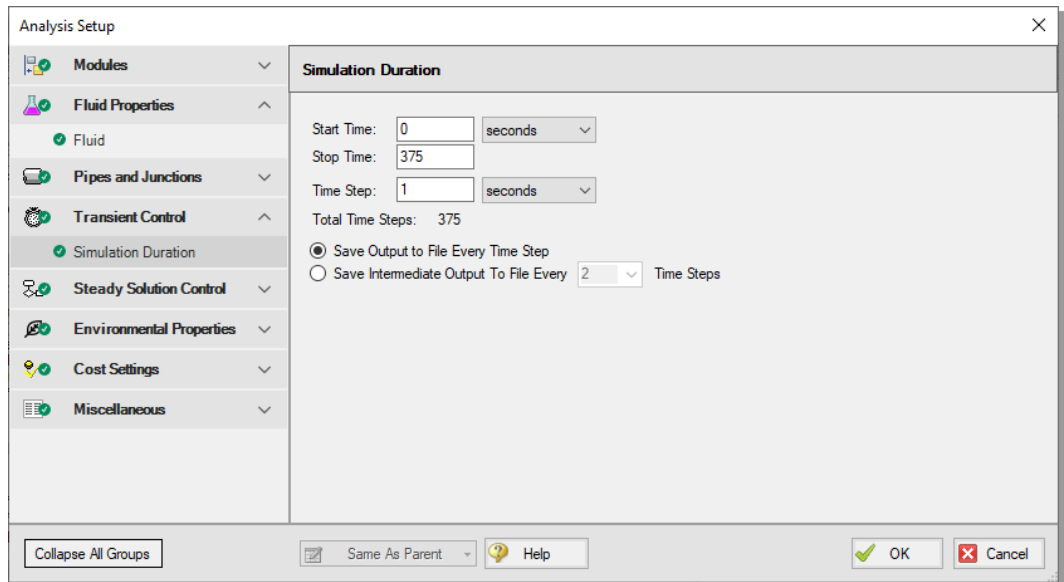

Figure 5.10   The Simulation Duration panel is used to control transient simulations

XTS Step 4. Specify fluid properties

1. In the Analysis Setup window on the Fluid panel, select the AFT Standard library, and then select "Air" in the fluids available window

2. Click "Add to Model" to select steam for use in this model

3. Select the "Redlich-Kwong" Equation of State option

4. Select the "Generalized" Enthalpy model

5. Click the OK button

XTS Step 5. Build the model

A. Place the pipes and junctions

This completes the first two groups of the Analysis Setup. The Pipes and Junctions group remains undefined. To complete this group, begin by assembling the model on the Workspace as shown in Figure 5.11.
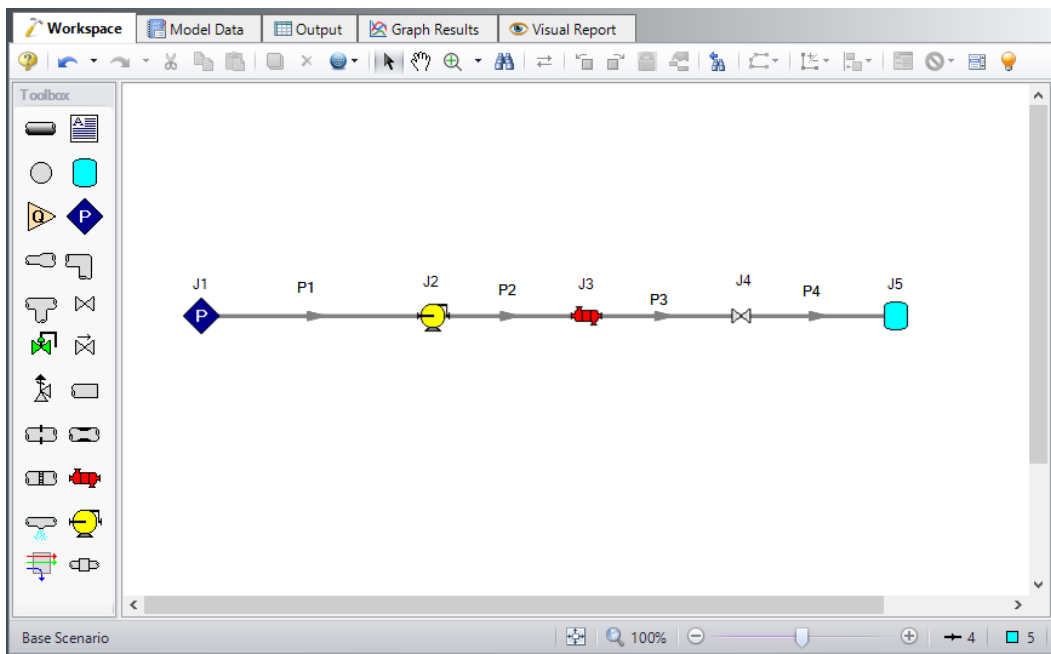

Figure 5.11   Workspace for Air Receiver Tank model

B. Enter the pipe data

The system is in place, but now you need to enter the input data for the pipes and junctions. Double-click each pipe and enter the following data in the Properties window.

All of the pipes are Steel – ANSI STD (Schedule 40).

Pipe	Length (meters)	Nominal Size
P1	6	3 inch
P2 - P4	3	3 inch

The AFT Arrow XTS module assumes adiabatic behavior for all pipes, so no heat transfer information is required for the pipes.

C. Enter the junction data

J1 Assigned Pressure Junction

1. Elevation = 0 meters

2. Fluid = Air

3. Stagnation Pressure = 0 barG

4. Stagnation Temperature = 21 deg. C

J2 Compressor Junction

1. Elevation = 0 meters

2. Compressor Model = Positive Displacement Compressor

3. Compression Process Thermodynamics = Polytropic

4. Polytropic Constant = 1.2

5. Compressor Sizing Parameter = Mass Flow Rate

6. Fixed Flow Rate = 170 sm3/h

Note: The polytropic constant entered for the compressor is intended to represent an active cooling element in the compressor.

J3 Heat Exchanger Junction

1. Elevation = 0 meters

2. Enter Loss Model Data as follows:

   a. Select the Loss Model as "Resistance Curve", and then click the "Enter Curve Data" button

   b. Select the "Flow Parameter" as "Mass" with units of "sm3/h"

    c.   Enter 170 sm3/h and 20 kPa in the Raw Data table

    d.   Click the "Fill as Quadratic" button to create data for a square loss curve

    e.   Select "Generate Curve Fit Now" to create a quadratic curve fit

3. Enter Thermal Data as follows:

    a.   Thermal Model = Controlled Downstream Temperature

    b.   Controlled Outlet Temperature = 30 deg. C

Note: The controlled downstream temperature is used to simplify the inputs for this example. For a more realistic analysis one of the cross-flow models should be used.

J4 Valve Junction

1. Elevation = 0 meters

2. Loss Model = User Specified

3. Subsonic Loss Model = Cv (ANSI/ISA)

4. Loss Source = Fixed Cv

5. Cv = 50

6. Xt = 0.7

J5 Tank Junction

The tank junction in the AFT Arrow XTS module can be modeled as finite in order to calculate the the changes in pressure, temperature, and mass in the tank over time. Define the air receiver tank using the Finite Tank Model as follows (Figure 5.12):

1. Tank Model = Finite

2. Initial Conditions

    a.   Pressure = 0 barG

    b.   Temperature = 21 deg. C

3. Finite Tank Conditions

a. Fixed Volume = 1900 liters

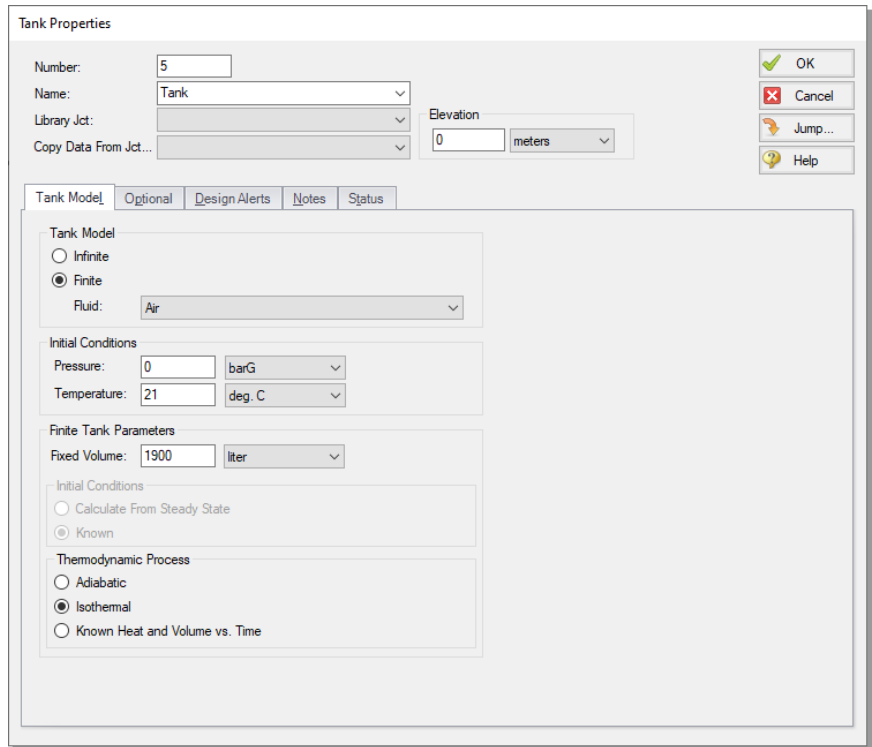b. Initial Conditions = Known

c. Thermodynamic Process = Isothermal

Figure 5.12   Properties window for the Tank junction defined as a Finite Tank

XTS 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 8.5 barG.

➢ **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 5.13.
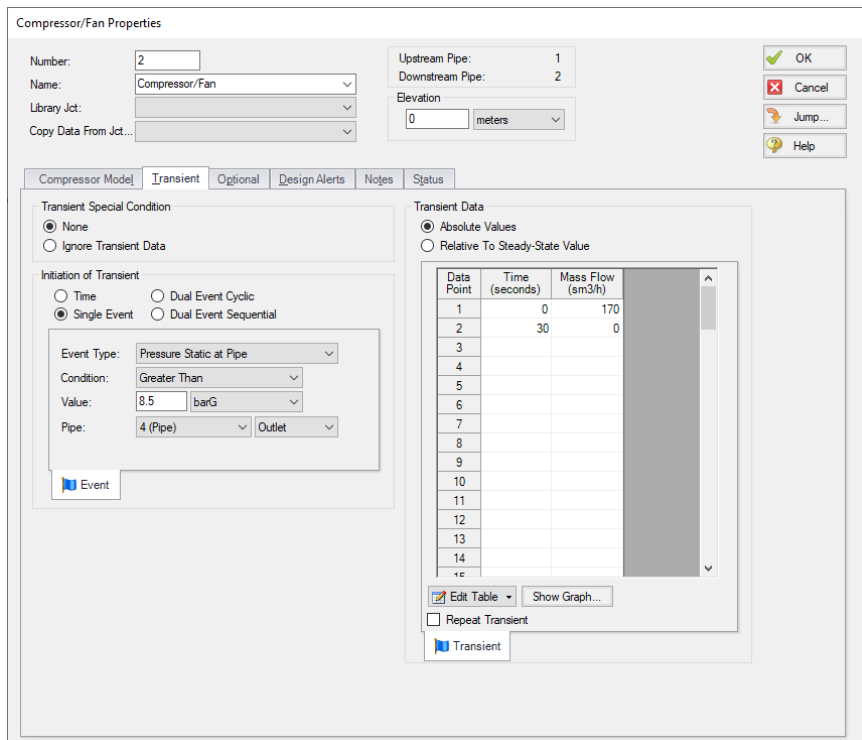

Figure 5.13   Compressor junction transient tab with flow transient defined

XTS 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.

After the run finishes, view the results by clicking the Output button.

XTS Step 8. Examine Transient Results

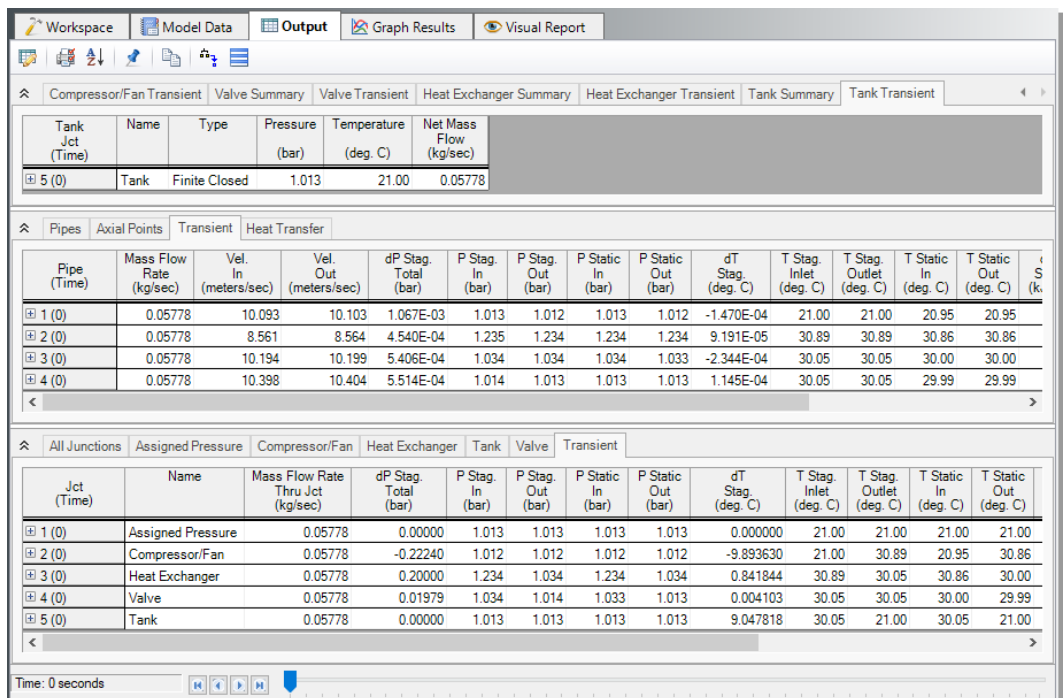AFT Arrow displays transient output for all Pipes and Junctions in the Output Window, as shown in Figure 5.14.

Figure 5.14 Transient output is displayed in the Output Window

Each of the summary tables in the General Output section has a companion transient summary tab which displays the summary data at each time step in the transient run. Each junction included in the summary is included in the transient summary. The transient summary data for each junction may be expanded or collapsed by clicking the + or – sign in beside the junction data list. The entire list may be expanded or collapsed by clicking the button in the top left-hand corner of the transient summary window. Figure 5.15 shows the Tank Transient summary tab, with all of the data sets collapsed. Figure 5.16 shows the Tank Transient summary tab, with the data for the J5 Tank expanded.

| Valve Transient | Heat Exchanger Summary | Heat Exchanger Transient | Tank Summary | Tank Transient |

Tank Jct (Time)	Name	Type	Pressure (bar)	Temperature (deg. C)	Net Mass Flow (kg/sec)
⊞ 5 (0)	Tank	Finite Closed	1.013	21.00	0.05778

Figure 5.15   Transient Summaries are displayed in the General Output section

| Valve Transient | Heat Exchanger Summary | Heat Exchanger Transient | Tank Summary | Tank Transient |

Tank Jct (Time)	Name	Type	Pressure (bar)	Temperature (deg. C)	Net Mass Flow (kg/sec)
⊟ 5 (0)	Tank	Finite Closed	1.013	21.00	0.05778
5 (1)	Tank	Finite Closed	1.040	21.00	0.05778
5 (2)	Tank	Finite Closed	1.065	21.00	0.05778
5 (3)	Tank	Finite Closed	1.091	21.00	0.05778
5 (4)	Tank	Finite Closed	1.117	21.00	0.05778
5 (5)	Tank	Finite Closed	1.142	21.00	0.05778
5 (6)	Tank	Finite Closed	1.168	21.00	0.05778
5 (7)	Tank	Finite Closed	1.193	21.00	0.05778
5 (8)	Tank	Finite Closed	1.219	21.00	0.05778

Figure 5.16   Transient Summary data can be expanded to show the junction data for every Time Step

The Event Messages tabs in the General Section (Figure 5.17) 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 333 seconds into the transient, meaning it took 333 seconds to pressurize the tank to the required pressure.
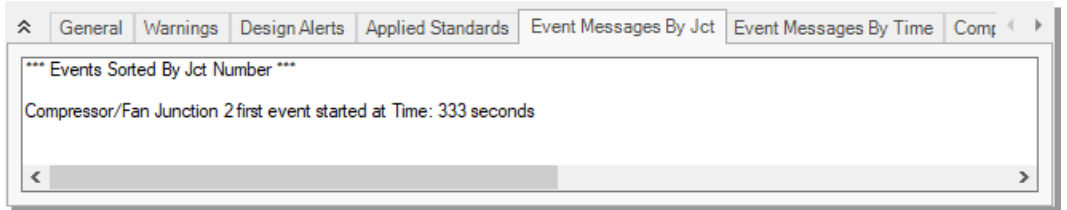

Figure 5.17   Event Messages tab in the General section of the Output window

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 5.18 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 9.9 bar once the compressor fully shuts off (Figure 5.19). The Graph Results window may also be used to create a full range of graphs related to the XTS module transient output.
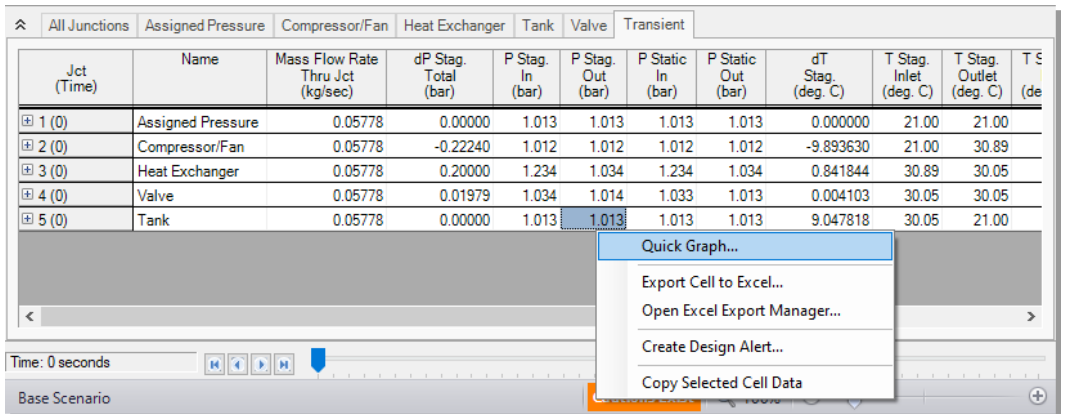

Figure 5.18   Use Quick Graph to easily plot J5 Tank pressure
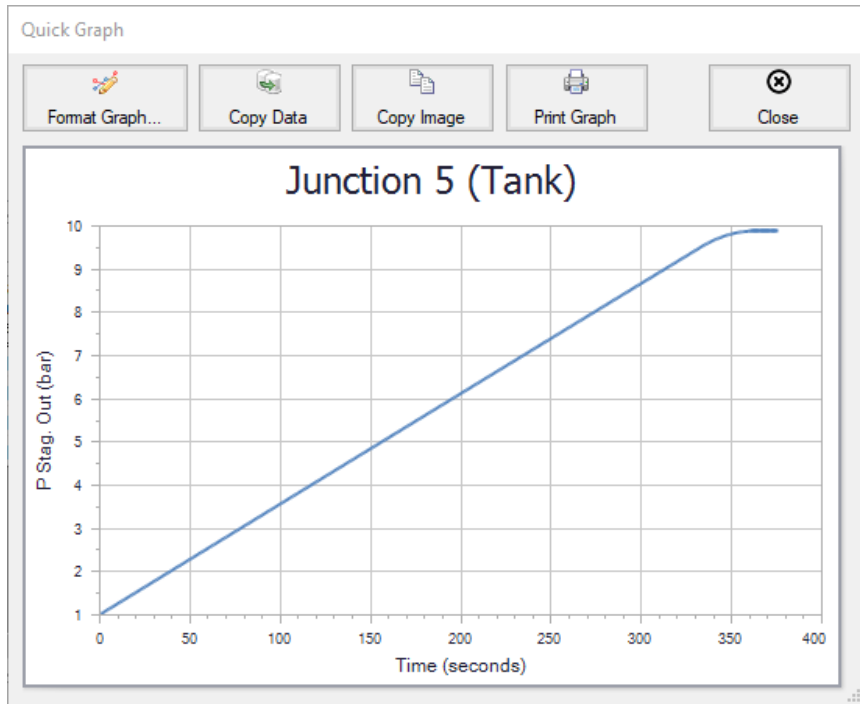
Figure 5.19   Quick Graph Plot of J5 Tank pressure

Similar to the summary transient data, the transient tabs in the Pipe and Junctions sections of the Output Window (shown in Figure 5.14) 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 5.20 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 5.21 shows the pipe and junction data at Time = 60 seconds.
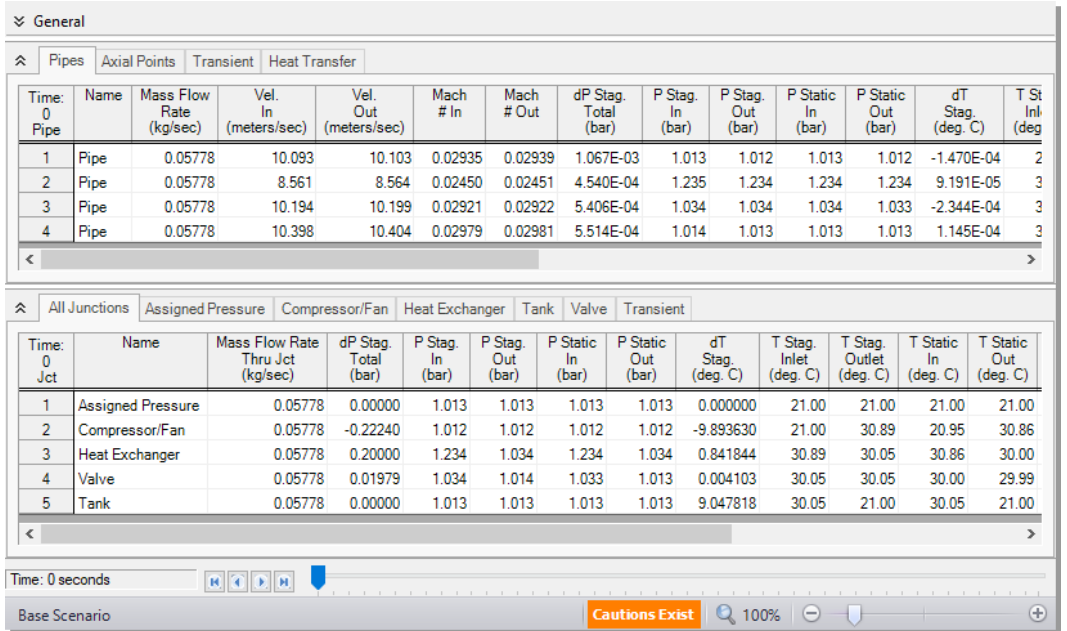
Figure 5.20   Pipe and Junction transient data at Time = 0 minutes

| Pipes | Axial Points | Transient | Heat Transfer |

Time: 60 Pipe	Name	Mass Flow Rate (kg/sec)	Vel. In (meters/sec)	Vel. Out (meters/sec)	Mach # In	Mach # Out	dP Stag. Total (bar)	P Stag. In (bar)	P Stag. Out (bar)	P Static In (bar)	P Static Out (bar)	dT Stag. (deg. C)	T St Inl (deg
1	Pipe	0.05778	10.093	10.103	0.02935	0.02939	1.067E-03	1.013	1.012	1.013	1.012	-1.470E-04	2
2	Pipe	0.05778	4.378	4.379	0.01173	0.01173	2.359E-04	2.760	2.760	2.760	2.759	3.570E-05	7
3	Pipe	0.05778	4.114	4.114	0.01179	0.01179	2.180E-04	2.560	2.560	2.560	2.559	5.351E-05	3
4	Pipe	0.05778	4.127	4.127	0.01183	0.01183	2.188E-04	2.551	2.551	2.551	2.551	4.524E-05	3

| All Junctions | Assigned Pressure | Compressor/Fan | Heat Exchanger | Tank | Valve | Transient |

Time: 60 Jct	Name	Mass Flow Rate Thru Jct (kg/sec)	dP Stag. Total (bar)	P Stag. In (bar)	P Stag. Out (bar)	P Static In (bar)	P Static Out (bar)	dT Stag. (deg. C)	T Stag. Inlet (deg. C)	T Stag. Outlet (deg. C)	T Static In (deg. C)	T Static Out (deg. C)
1	Assigned Pressure	0.05778	0.000000	1.013	1.013	1.013	1.013	0.000000	21.00	21.00	21.00	21.0
2	Compressor/Fan	0.05778	-1.747806	1.012	1.012	1.012	1.012	-53.506438	21.00	74.51	20.95	74.5
3	Heat Exchanger	0.05778	0.200000	2.760	2.560	2.759	2.560	44.498090	74.51	30.01	74.50	30.0
4	Valve	0.05778	0.008105	2.560	2.551	2.559	2.551	0.001662	30.01	30.01	30.00	30.0
5	Tank	0.05778	0.000000	2.552	2.552	2.552	2.552	9.006653	30.01	21.00	30.01	21.0

Time: 60 seconds

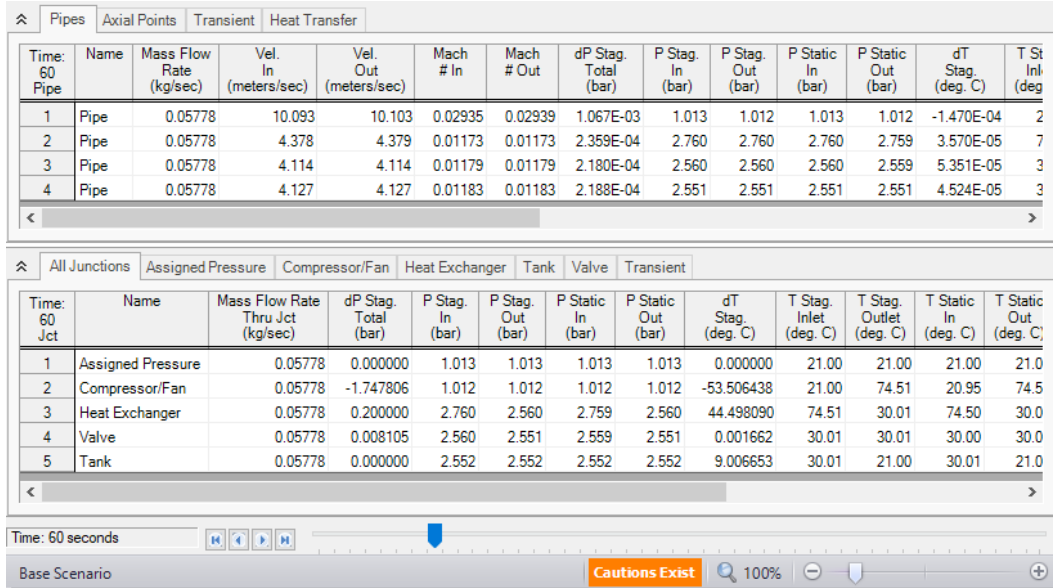Base Scenario   Cautions Exist   100%

Figure 5.21   Pipe and Junction transient data at Time = 60 seconds

Other AFT Arrow Capabilities

This Quick Start Guide necessarily omitted coverage of a number of AFT Arrow capabilities. This chapter briefly describes some of the important capabilities not covered.

Microsoft Excel™ data integration

AFT Arrow now includes enhanced Excel importing and robust Excel exporting. An Excel spreadsheet can be used to vary selected input parameters in multiple scenarios. Selected output can also be exported to specified Excel sheets and cells.

Integration with other software and data standards

AFT Arrow includes a number of importing and exporting capabilities. Piping layouts and dimensional data can be imported from GIS Shapefiles (SHP) to build a model. Additionally, CAESAR II Neutral Files (CII) can be imported into AFT Arrow and EPANET Files (INP) can be both imported and exported.

Piping Component Files (PCF) from AutoCAD Plant 3D, SmartPlant, PDS, CADWorx, and other software can be imported into AFT Arrow. These options are all accessed from the File Menu.

Gas mixtures using NIST REFPROP and Chempak

NIST REFPROP and Chempak support gas mixing in two forms. First, users can create their own mixtures. These are called pre-mixtures

because they are defined before the model is run. Second, as flow streams of different composition mix, an entirely new gas mixture results. Its composition depends on the composition of the incoming streams as well as the relative flow rates. Since in general the flow rates are not known by the user, AFT Arrow updates the composition of this new mixture as the simulation progresses. For this reason these mixtures are referred to as dynamic mixtures. AFT Arrow is restricted to non-reacting gas mixture calculations.

The ASME Steam Tables library

The ASME Steam Tables Library offers users access to fluid properties obtained from the IAPWS Industrial Formulation 1997 for the Thermodynamic Properties of Water and Steam (ISPWS-IF97). This library is selected by choosing the option at the top of the Fluid panel in Analysis Setup, at which point it is automatically displayed in the Fluids in Current Model area.

Separator junction

AFT Arrow supplies a separator junction. This junction type allows you to separate fluids out of a mixture. Since mixtures can only be modeled using Chempak, this junction requires Chempak.

Fitting library

AFT Arrow offers a library of about 400 fitting losses that can be added to pipes.

Network libraries

Junction components and pipe materials can be saved to libraries for later reuse. Libraries can be located on local PC's or deployed across local or wide area networks. The Library Manager allows users to connect to relevant libraries for their specific pipe system design.

Varied pipe geometries including rectangular duct

Five different pipe geometries can be modeled in AFT Arrow. The default geometry is the cylindrical pipe. The other four geometries are rectangular duct, cylindrical annulus, noncylindrical pipe, and helical tube.

Sonic choking

AFT Arrow models sonic choking in complex pipe systems and accurately predicts the resulting choked flow rate. In addition, AFT Arrow can accurately calculate multiple choking points in the system.

Rotating systems for turbomachinery

AFT Arrow allows users to specify rotational velocities such as those experienced in turbomachinery. AFT Arrow uses this information to determine the varying acceleration body force on each individual pipe as determined by the laws of rotational dynamics.

Elevation changes

AFT Arrow uses elevation differences between junctions to calculate hydrostatic or gravity pressure changes in a system. This can be especially useful for high density gases with moderate elevation changes, pipe systems with significant elevation changes such as those found in overland natural gas pipelines, and gas piping for deep mining operations.

Relief system modeling

AFT Arrow provides support for modeling relief systems, including relief valve junctions and support for ANSI/ISA Cv loss coefficients. Sonic capabilities allow modeling of high velocity conditions often found in relief systems.

Control valve modeling

AFT Arrow can model four types of control valves. They are Pressure Reducing Valves (PRVs), Pressure Sustaining Valves (PSVs), Flow Control Valves (FCVs), and Pressure Drop Control Valves (PDCVs).

Variable speed compressors and fans

AFT Arrow allows you to model controlled compressors or fans where the control parameter is flowrate, discharge, or suction pressure. The compressor or fan speed is varied to deliver the flow rate or discharge pressure you set.

Heat of compression

AFT Arrow allows users to choose several different compression models for compressor junctions. The compression process models how the heat of compression affects the gas temperature, which is especially important in compressor applications.

Insulation on inside of pipe/duct

Insulation can be placed on the inside of a pipe or duct, and the effect of reduced flow area on the pressure drop will be automatically determined. If modeling heat transfer, the effect of the insulation on the inside will be accounted for.

Data varied through Microsoft Excel™

You can use an Excel spreadsheet to vary selected input parameters.

Model heat transfer between loops

Heat transfer between loops which are hydraulically isolated can be modeled. The loops can have different fluids. This is called thermal linking.

View input and output data from different scenarios concurrently

Input data from different scenarios can be viewed concurrently in the Model Data window. Similarly, output data from different runs can be viewed concurrently in the Output window.

Design Alerts

Design Alerts for one or more parameters can be entered for a pipe. A common use is maximum and minimum allowed operating pressure. If the resulting value for that parameter violates the Design Alert a warning message is given in the output.

Cost calculations

Material, Installation and Maintenance costs can be calculated for junctions, pipes and fitting based on Cost Libraries which are user-created. In addition, energy (operation) costs can be calculated for compressors/fans. Time value of money can be taken into consideration by entering inflation and interest rates. The results are presented in a Cost Report in the Output.

Retaining graphs

Comprehensive graphing features allow users to graph a wide variety of parameters at single points or along flow paths within their model. Graphs added to a Graph List are saved so that the graph can be regenerated if changes are made to the model input, or if the user desires to plot the same graph in multiple scenarios. Multiple graphs can be shown at the same time.

Index

V

W

AFT Arrow Add-on Modules

ANS

Automated Network Sizing (ANS)
Evaluates the complex interaction of system variables to reveal combinations of cost savings

XTS

Extended Time Simulation (XTS)
Models dynamic system behavior and how critical system parameters vary over time.

GSC

Goal Seek & Control (GSC)
Identifies input parameters that yield desired output values and simulates control functions

For Product Support:
Call: (719) 686-1000
Email: support@aft.com

AFT
Applied Flow Technology