Chemical Engineering 2O04
Introduction to Fluid Mechanics
Pipe-Flo: Advantage Student Edition 17 **Tutorials**

Custom Courseware

Ryan J. LaRue, *B.Eng.Mgmt*
David R. Latulippe, *Ph.D*

Revision: Winter 2021

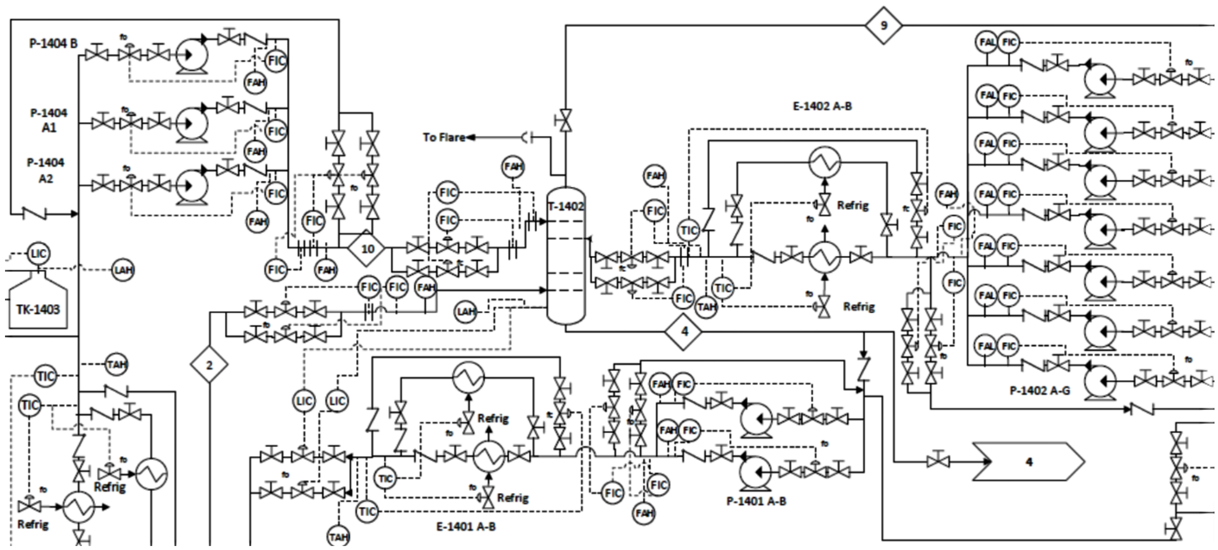
The authors wish to acknowledge the following individuals for their role in developing these course materials:

Pedro de Oliveira Filho for creating the first version of the PIPE-FLO courseware, thus laying the groundwork for subsequent editions.

Ashleigh Warren for updating the courseware to its current state which services the latest version of the PIPE-FLO software.

Contents

Introduction to Pipe-Flo



A small portion of a process and instrumention diagram for a large hypothetical syngas processing plant. Given the complexity and size of the facility, any attempt to design or analyze by hand a significant portion of this piping network would prove to be infeasible.

Pipe-Flo is a leading fluid flow design and modeling software which allows users to efficiently simulate and analyze complex fluid transport systems. In almost every workplace that requires a chemical engineer, there is a fluid distribution system, in one form or another. Therefore, it is important for the modern chemical engineer to have the tools to design new fluid distribution systems or to troubleshoot and maintain existing networks. In the real world, piping systems are extraordinarily complex. While manual procedures—like the Hardy-Cross Method—for determining flow in piping systems do exist, solving many real-world problems with these procedures is *practically impossible*. By easily calculating factors such as pressure losses, pump requirements, and flow rates, a fluid flow design and modeling package like Pipe-Flo is a indispensable tool.
To run Pipe-Flo, you will require:

- A PC with Microsoft Windows 8 (or newer) installed. *Mac OS and Windows XP are not supported*;
- At least 4 GB of System RAM;
- At least 200 MB of hard drive storage;
- 1366×768 screen resolution; and
- An internet connection.

The sets of tutorials presented in this courseware have been designed to help ChemEng 2O04 students learn the basics of the Pipe-Flo software. If you find that extra resources are required, the makers of Pipe-Flo (*Engineered Software*) offer free tutorials on their website at http://kb.eng-software.com/eskb/pipe-flo. Use these resources to your full advantage to get the most out of this software tool.

Tutorial #1:
Learning the Basics

Tutorial #1 will consider the following concepts:

- Creating and customizing new Pipe-Flo projects
- Features of the Pipe-Flo interface
- Navigating the Pipe-Flo interface
- Interpreting simulation error messages

1.1 Creating and Customizing Pipe-Flo Projects

Opening the Pipe-Flo program from your desktop or start menu will display the window shown in Figure 1.1. The central panel features useful information such as links to recent files and support documentation; scroll down to access features such as training videos, software manuals, pump catalogs, and other useful tools.

Pipe-Flo allows users to customize their workspace. Start by creating a *"New Project"*, which opens the screen seen in Figure 1.2. This can be accomplished by either:

- Clicking File › New in the drop-down menu at the top of the screen; or
- Typing Ctrl + N.

Units. In this course (as well as in your future jobs), you will be required to analyze problems in different systems of units. Helpfully, Pipe-Flo is capable of handling different systems of units. You can customize how quantities are input and displayed by accessing the *"Adjust Document Units"* dialog box, in one of two ways, as seen in Figure 1.3:

The user can then customize several different aspects of the workspace, including the following:

- Select System › Units from the menu at the top of the screen; or
- Click the shortcut icon (UNITS) located in the center of ribbon.

The *"Adjust Document Units"* dialog box is shown in Figure 1.4; Pipe-Flo displays *non-SI units* by default. For example, pressure is displayed in units of pounds per square-inch, power is given in horsepower, lengths are given in feet, densities are given in pounds per cubic foot, and temperatures are shown in degrees Fahrenheit. Use this dialog box to change the displayed units for your project; click OK to finish. *It is strongly recommended that you set your default document units to the standard SI/metric units.*[1]

[1] Non-default units can be selected manually, at any time.

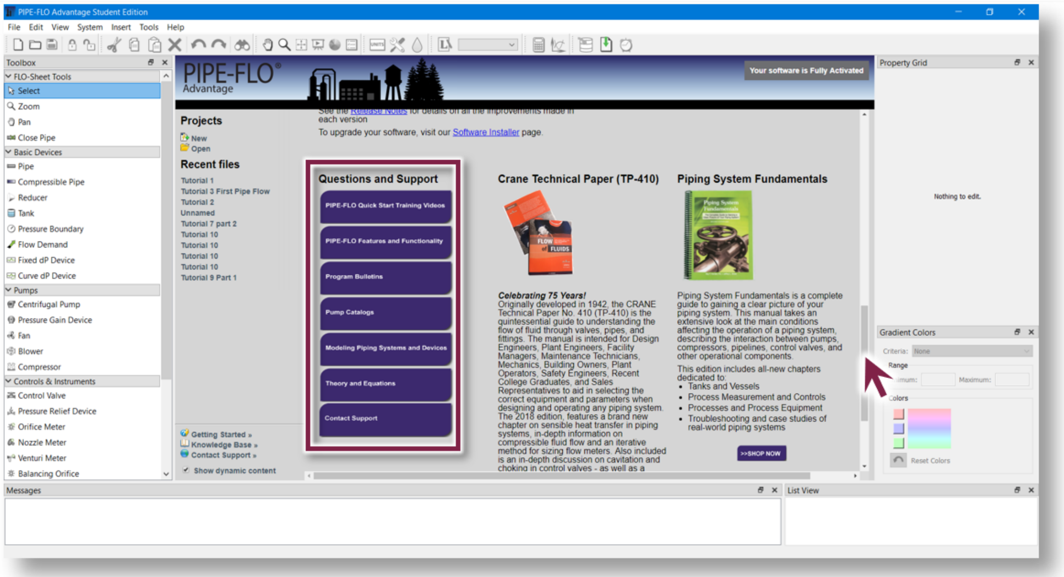


Figure 1.1: *The home screen of the* Pipe-Flo *interface; scroll down to find useful links to training videos, features, program bulletins, real-life pump catalogs, fluid mechanics modeling and theories, as well as program support.*

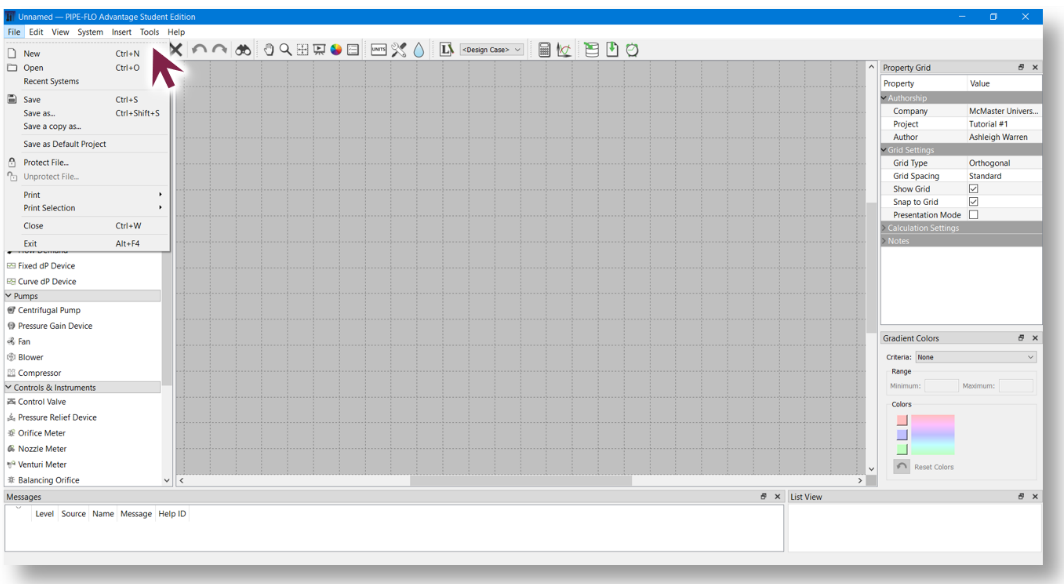


Figure 1.2: *Creating a blank* Pipe-Flo *workspace in a new project.*

It is critical to note that by default, Pipe-Flo utilizes a *gage (gauge) pressure reference* as oppose to an *absolute pressure reference*. In these tutorials, values are *always given in terms of absolute pressure*. It is highly-recommended that you set the pressure reference to “absolute”, as seen in Figure 1.4. Similarly, you can choose the basis of flow: volumetric, mass, or standard. For now, we will assume that flow will be on a volumetric basis.

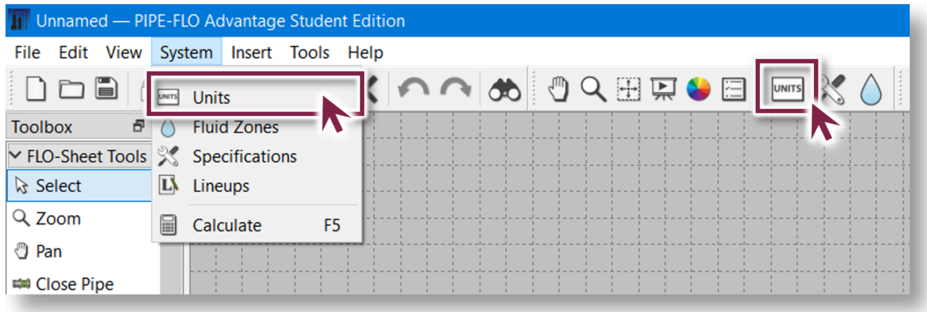


Figure 1.3: *Two analagous methods of accessing the "Adjust Document Units" dialog box.*

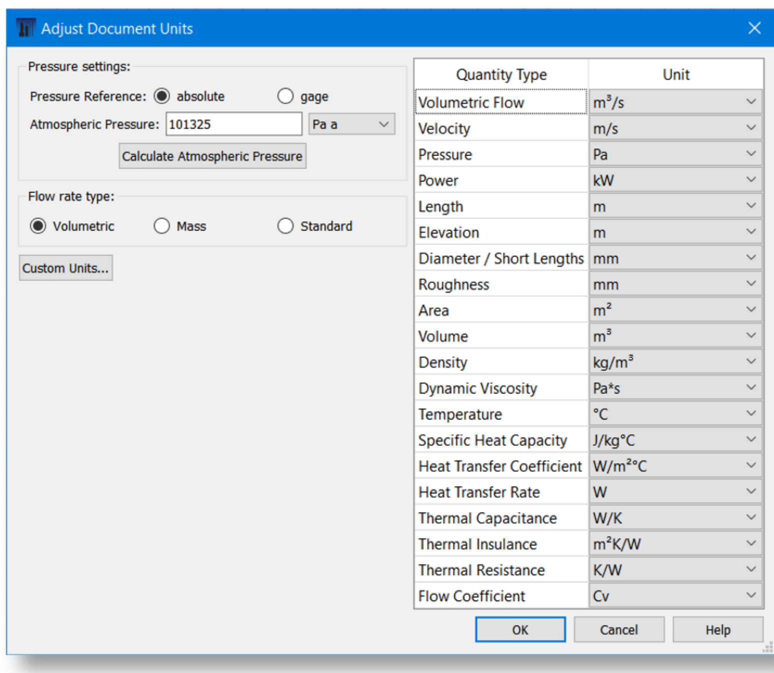


Figure 1.4: *Overriding the default (Imperial) display units. It is recommended that you use the settings shown here.*

Simulation Properties. You can use the *"Property Grid"*—docked on the right-hand side of the interface—to manipulate the properties of your document, including the document name and author, grid settings, calculation settings, and document notes. Click on the "❯" or "✔" symbols on the headings in the *Property Grid* to access or hide the additional settings. Figure 1.5 demonstrates how to set the "authorship" of a PIPE-FLO simulation.

Scrolling downwards, you can manipulate the *"Grid Settings"* of the FLO-SHEET—the gray workspace in the center of your screen. Clicking on each setting will open a description of that setting, but if no description is displayed, this can be corrected by *dragging upwards* the border of the window directly below the *Property Grid*, with the resizing pointer (↔). This can be seen in Figure 1.6. The width of the windows—as well as the FLO-SHEET—can be resized in a similar fashion.

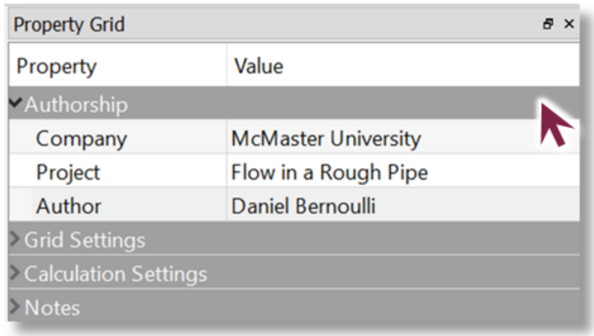


Figure 1.5: *Specifying the company, project name, and author.*

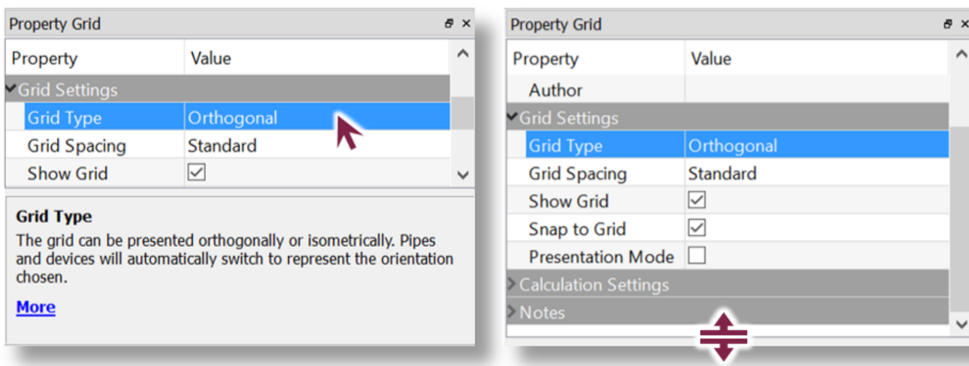


Figure 1.6: **(L)** Modifying the grid settings of the Flo-Sheet. A description of each setting can be displayed below the Property Grid. **(R)** The description window can be unhidden by hovering the cursor over the window border, and resizing upwards.

Scrolling further downwards, the *"Calculation Settings"* can be manipulated, including parameters such as:

- The method used to calculate frictional losses;
- The convergence tolerance for iterative calculations;
- The maximum possible number of iterations;
- The critical Reynold's number (Re) between the laminar and mixed/turbulent flow regimes; and
- A provision to force Pipe-Flo to assume laminar flow.

This can be seen in Figure 1.7. In general, these calculation settings can be left as-is. For further information on each of these parameters, the description window can be unhidden, as explained previously.

We will always use the *Darcy-Weisbach* calculation method, a *Laminar Flow Cutoff Re* of $Re = 2300$, and an atmospheric pressure of 101.325 kPa (absolute), unless otherwise noted! Be sure to set these parameters, as such!

Finally, you can add your own notes to the simulation file under the *"Notes"* heading at the bottom of the *Property Grid*. Clicking on the *Notes* field will bring up a dialog box, as seen in Figure 1.8. under the *Edit* tab, any simulation notes can be added. Click OK to finish.

After you create a modified workspace, do not forget to save your preferences! By saving your document as *template*, you will be able to load all your customized units and settings each time a new document or project is created. This can be done by clicking File › Save as Default Project, as seen in Figure 1.9.

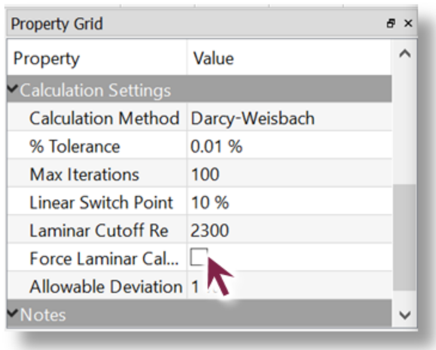


Figure 1.7: *Modifying the calculations parameters.* PIPE-FLO *can be forced to assume laminar flow in all cases if the "Force Laminar Calculation" checkbox is ticked. Ensure that this setting remains* disabled.

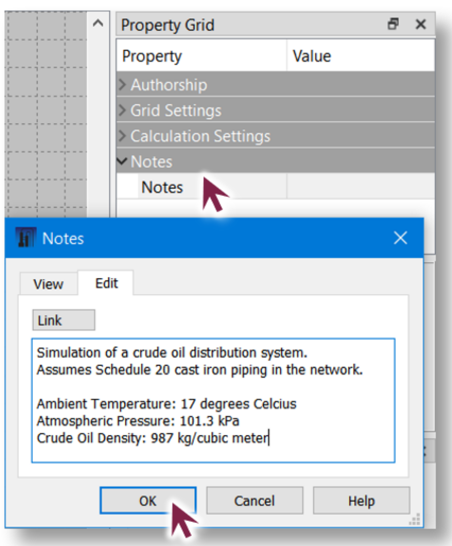


Figure 1.8: *Adding simulation notes to the document.*

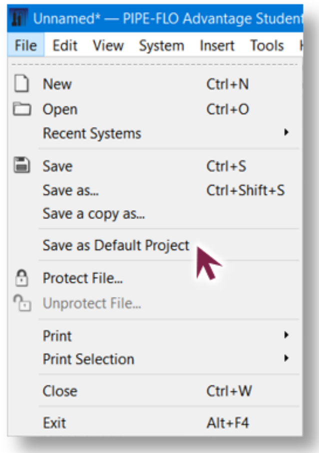


Figure 1.9: *Saving a workspace as the "default project" allows the user to utilize that workspace's settings in subsequent workspaces.*

1.2 Navigating the Pipe-Flo Interface

This section will introduce the different features of the Pipe-Flo interface which are used in the modeling and simulation of piping systems.

Ribbon Icons. The ribbon at the top of the screen—as seen in Figure 1.10—contains many useful shortcuts and features. The following subsection is an overview of the each of these icons and their respective functions:

1. The standard *"New Document"*, *"Open"*, and *"Save"* icons. As is customary, shortcut key commands (*i.e.* Ctrl+N, Ctrl+O, and Ctrl+S) perform the same functions.
2. *"Password Protect this File"*: allows the user to require a password before the file will open. Caution! If you lose the password, the file cannot be retrieved.
3. *"Remove Password Protection from this File"*: allows the user to cancel the password protection, but the user must first enter the password to do so.
4. The standard *"Cut"*, *"Copy"*, *"Paste"* and *"Delete"* icons.
5. The standard *"Undo"*, and *"Redo"* icons.
6. *"Find Device by Name"*: allows the user to search for a device on their flow sheet. This can also be achieved by using Ctrl+F.
7. The standard *"Pan"*, and *"Zoom"* icons. With the *Zoom* mode enabled, to *zoom in*, click the *left mouse button*. to *zoom out*, click the *right mouse button*. Scrolling up and down with the mouse wheel also achieves the same effect.

Figure 1.10: *The* Pipe-Flo *ribbon.*

8. *"Zoom to Extents"*: automatically chooses the zoom which best fits and centers the entire FLO-SHEET on the user's screen.

9. *"Presentation Mode"*: this toggle button temporarily removes the gridlines from the screen, and changes the background color to white. This option makes the FLO-SHEET appear less busy and more suitable for printing and presenting.

10. *"Disable Colors"*: this toggle button reverts certain colors to grayscale, and temporarily removes gradient coloring.

11. *"Device View Options"*: this useful option allows the user to choose which properties will be displayed directly on the FLO-SHEET next to their respective unit operations (*i.e.*tanks, pumps, pipes, etc.).

12. *"Units"*: allows the user to modify the default units for the project, as well as other parameters. This feature was described on Page 1.

13. *"Specifications"*: allows the user to specify the properties of the pipes to be used in the simulation. PIPE-FLO offers a large range of piping options, with pre-loaded parameters including the piping material and schedule, the roughness value, and the exact pipe diameter. The user may also define their own pipes.

14. *"Fluid Zones"*: allows the user to specify the types of fluids to be used in the simulation. PIPE-FLO collects fluid data such as viscosity, density, vapour pressure, and heat capacity for many different fluids over a wide range of temperature/pressure conditions. PIPE-FLO also allows users to define their own fluids.

15. *"Edit Lineups"*: allows the user to specify and save multiple independent operating conditions to be simulated.

16. *"Set Current Lineup"*: allows the user to choose which lineup—previously defined under *Edit Lineups*—is to be simulated.

17. *"Enter/Exit Calculation Mode for Document"*: this toggle button turns on or off the "calculation mode". Typing F5 also performs the same function. In "calculation mode", PIPE-FLO will simulate the FLO-SHEET and display the calculated outputs along with each device. Provided that there are no compilation or simulation errors, this will happen automatically if calculation mode is "on". Switching calculation mode to "off" prevents the software from displaying errors while the FLO-SHEET is developed or edited.

18. *"Graph"*: this feature will graph specific results of a simulation for a process unit. The applicable unit (a pipe, pump, etc.) must first be selected (highlighted), and calculation mode must be turned on. Typing Ctrl+G also activates this mode.

19. *"Send to DataLink"*: exports data to an Open Database Connectivity (ODBC). Essentially, PIPE-FLO will upload data and simulation results to an external database. This feature is only available in fully-licensed copies of the software.

20. *"Data Import"*: imports data from a specified source to be used in the simulation. This feature is also only available in fully-licensed copies of the software.

21. *"Overtime Simulation"*: enables you to view changes in your piping system as they occur during a simulation. We will not use this feature in the tutorials.

In addition to the shortcut icons on the ribbon, these commands can all be accessed through the menus at the top of the screen.

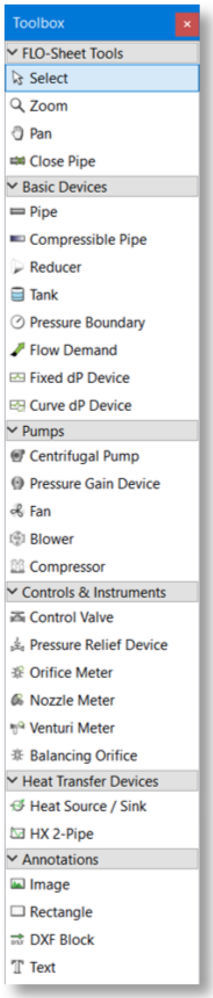


Figure 1.11: *The* PIPE-FLO *toolbox.*

The Toolbox. The PIPE-FLO *"Toolbox"* is docked to the left-side of the workspace. It is a collection of the FLO-SHEET tools, basic devices, pumps, controls and instruments, heat transfer devices,[2] and process annotations that are available to the user. The *Toolbox* is pictured in Figure 1.11.

The following summary contains a brief description of each the contents of each of the headings in the *Toolbox*. Use the "❯" or "⌄" symbols on the headings to access or hide the tools.

- FLO-SHEET *Tools*: the *"Select"* cursor allows the user to select and modify process units in the FLO-SHEET. The *Zoom* and *Pan* tools are identical to the ones found on the ribbon. When the *"Close Pipe"* tool is enabled and a pipe segment is selected, this feature manually forbids fluid flow in that segment. Clicking the pipe segment again re-opens the pipe.
- *"Basic Devices"*: this heading contains simple process units such as pipes and tanks. Also found within the *Basic Devices* heading are simplified process units such as the *"Pressure Boundary"*, *"Flow Demand"* and pressure-changing (*"dP"*) devices.
- *"Pumps"*: various types of pumps can be found under this heading.
- *"Controls & Instruments"*: various types of valves, metering devices and orifices can be found under this heading.
- *"Heat Transfer Devices"*: generic heat sources/sinks, and simple two-pipe heat exchangers can be found under this heading.
- *"Annotations"*: these features allow the user to add images, draw rectangles, and custom text to the FLO-SHEET. *Annotations* do not affect FLO-SHEET calculations and results.

With exception to the "FLO-SHEET *Tools*", each unit can be added to the FLO-SHEET by clicking on the unit's icon and then clicking on the FLO-SHEET.

List View. Generally docked towards the bottom (right) of the workspace, the *"List View"* summarizes the simulation results for each process unit and pipe in the FLO-SHEET. Figure 1.12 highlights some of the key features of the *List View*.

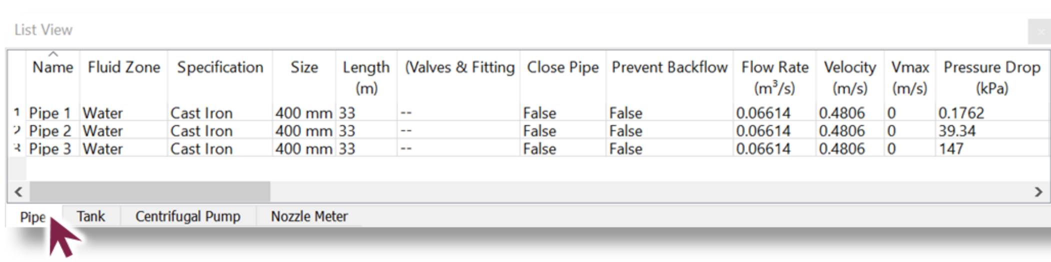


Figure 1.12: *The List View window displays the results of the converged simulation, including values such as flow rates, pressure drops, fluid velocities Reynold's numbers (Re), and if backflow occurs.*

[2] Heat transfer in systems with fluids—formally introduced in Chemical Engineering 3A04, Introduction to Heat Transfer—can also be modeled in PIPE-FLO.

Error Messages. The *"Messages"* window—seen in Figure 1.13—is generally docked in the bottom left corner of the PIPE-FLO workspace. It indicates errors in the simulation, any pertinent cautionary notes, as well as additional information related to simulation convergence. Error messages will be covered in detail in §1.3.

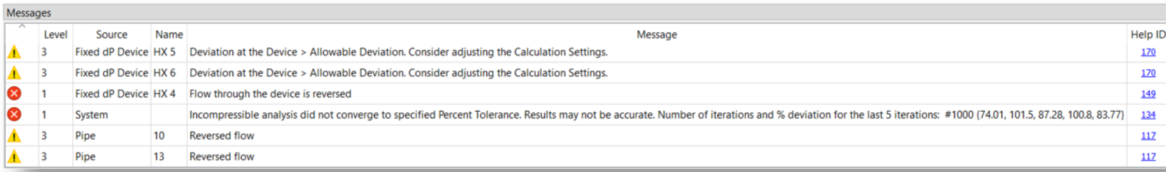


Figure 1.13: *The Messages window indicates the presence of simulation errors, and offers additional pertinent information.*

Gradient Colors. This PIPE-FLO feature helps users to visualize various flow conditions. Users can apply color gradients to the flowsheet to convey information regarding properties such as flow rates, pressures, elevation changes, head losses, fluid temperatures, or flow velocities. You can apply a color gradient to a particular property by accessing the *"Gradient Colors"* window docked in the bottom (left) corner of the screen. Figure 1.14 illustrates the effect of using the *Gradient Colors* feature to visualize the pressure drop over the length of a pipe segment.

Navigation. Each of the windows that were previously mentioned are said to be *dockable*. This means that they can be unfixed from the interface, dragged to any position and/or be resized. Using the restore icon (❐) allows the user to undock and drag each of the windows to any location in the workspace. *Undocked* windows can be resized by hovering your cursor over the borders, then clicking and dragging when the "⟷" cursor appears. In a similar fashion, the "⇹" cursor can be used to resize *docked* windows. Undocked windows can be redocked again by dragging them to their preferred location, and releasing them once the docking location is highlighted in blue.

> Should you (accidentally) close a window, it can be easily restored later. Navigating to View › ⟨Window⟩ will restore any closed ⟨Window⟩ of interest. Or, the key shortcuts Ctrl + 1 – 6 can be typed to open or close the corresponding windows. Similarly, navigating to View › Toolbars › ⟨Toolbar⟩ can be used to show or hide a ⟨Toolbar⟩ in the *Toolbox*.

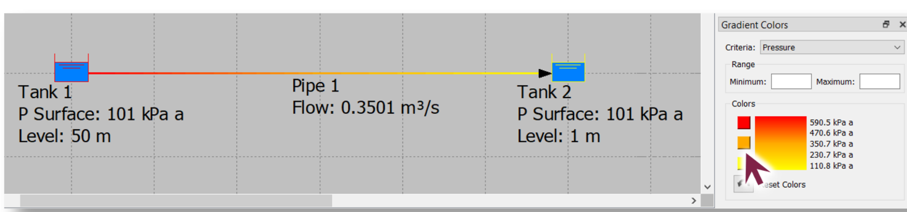


Figure 1.14: *A color gradient portrays the decrease in pressure along a length of pipe connecting two tanks of differing elevations.*

1.3 Error Messages

The PIPE-FLO *Messages* window displays any messages, warnings, and errors that may be pertinent to the user. These messages are categorized into five levels of decreasing severity, starting at *Level 0*. According to the makers of PIPE-FLO, these alert levels correspond to:

Level 0 Pre-calculation check or calculation engine errors are preventing the program from calculating any results (*e.g.* physically-impossible parameters entered, like a negative absolute pressure). If a Level 0 message is generated, no other level messages will be displayed because no results have been calculated.

Level 1 Even though results may be calculated, the results are invalid or a system configuration is invalid (*e.g.* reversed flow or missing parameters). The results are unequivocally wrong and require close evaluation.

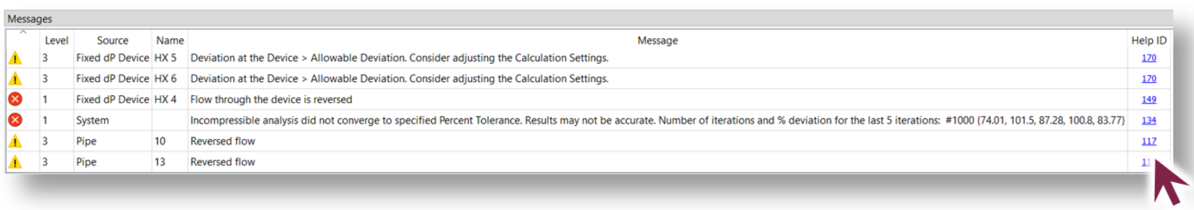
Level 2 The calculated results exceed either the limits of the equipment or limits of the equations used to obtain the results (*e.g.* the calculated flow rate through a pump exceeds the range for which data is available). There is increased uncertainty with regards to the accuracy of the displayed results.

Level 3 The calculated results exceed specified operational limits (*e.g.* the pressure in a tank exceeds its design limits).

Level 4 This level constitutes informational messages that may or may not be based on the calculated results. These messages provide information about the status of devices (*e.g.* the number of iterations, the validity of laminar flow assumptions).

As seen in Figure 1.15, clicking on the message in the *Messages* window will highlight the offending device and load its settings into the *Property Grid*. Clicking on the hyperlink under *"Help ID"* will open a help page which will attempt to explain that particular error.

For additional Pipe-Flo resources, the program documentation can be accessed by navigating to Help › Contents, or by typing F1.


Messages

	Level	Source	Name	Message	Help ID
	3	Fixed dP Device	HX 5	Deviation at the Device > Allowable Deviation. Consider adjusting the Calculation Settings.	170
	3	Fixed dP Device	HX 6	Deviation at the Device > Allowable Deviation. Consider adjusting the Calculation Settings.	170
	1	Fixed dP Device	HX 4	Flow through the device is reversed	149
	1	System		Incompressible analysis did not converge to specified Percent Tolerance. Results may not be accurate. Number of iterations and % deviation for the last 5 iterations: #1000 (74.01, 101.5, 87.28, 100.8, 83.77)	134
	3	Pipe	10	Reversed flow	117
	3	Pipe	13	Reversed flow	11

Figure 1.15: *The Messages window indicates the severity of simulation errors, denotes the device that is causing the error, and provides a description of possible causes of the error.*

Tutorial Problems

Problem #1. Create a new document with the following settings:

(A) Set the "Company" to be McMaster University, the 'Project" to be "Tutorial #1", and the "Author" to be your name.

(B) Ensure that the calculation method is "Darcy-Weisbach", and the tolerance is 0.01%.

(C) Set the "Laminar Cutoff *Re*" (Reynold's Number) to be 2300.

(D) All units should be displayed as SI units, or as units based on the metric system (*e.g.* kW, mm, °C, etc.) where standard SI units are unavailable or inconvenient. Note: the *"Flow Coefficient"* should remain as "*Cv*" (C_v).

(E) The pressure reference should be "absolute" and the atmospheric pressure should be set to 101.325 kPa (absolute).

(F) Enable the "Snap to Grid" mode.

⚠ Save this project that you just created as your *default document*. Note that any new document that is created subsequently will have the same initial settings as this default document! Unless otherwise indicated, all the subsequent tutorials in this courseware will use this default document as the basis for new projects.

Problem #2. To which window or dialog box would you navigate in order to access the following features? Check all that apply.

(A) Determine the viscosity of cyclohexane at 50°C and 200 kPa.

(B) Display mass flow rates instead of volumetric flow rates.

(C) Add an orifice meter to the FLO-SHEET.

(D) Choose "Stainless Steel 316" to be the piping material for a particular segment.

(E) Set the liquid level in a tank.

(F) View the liquid level in a tank.

(G) Determine the number of iterations needed to obtain the simulation results.

(H) Visually identify particular pipes with high head losses.

(I) Increase the maximum number of simulation iterations to 1,000.

(J) Find the roughness coefficient for a cast iron pipe.

(K) Choose ethanol as one of the process fluids.

(L) Investigate why the simulation fails to produce a result.

Feature	*Property Grid*	*Toolbox*	*List View*	*Lineup Values*	*Messages*	*Gradient Colors*	*Units*	*Fluid Zones*	*Specifications*
(A)	☐	☐	☐	☐	☐	☐	☐	☐	☐
(B)	☐	☐	☐	☐	☐	☐	☐	☐	☐
(C)	☐	☐	☐	☐	☐	☐	☐	☐	☐
(D)	☐	☐	☐	☐	☐	☐	☐	☐	☐
(E)	☐	☐	☐	☐	☐	☐	☐	☐	☐
(F)	☐	☐	☐	☐	☐	☐	☐	☐	☐
(G)	☐	☐	☐	☐	☐	☐	☐	☐	☐
(H)	☐	☐	☐	☐	☐	☐	☐	☐	☐
(I)	☐	☐	☐	☐	☐	☐	☐	☐	☐
(J)	☐	☐	☐	☐	☐	☐	☐	☐	☐
(K)	☐	☐	☐	☐	☐	☐	☐	☐	☐
(L)	☐	☐	☐	☐	☐	☐	☐	☐	☐

Problem #3. Brainstorm a few examples of situations that might cause each "level" of error that can be found in the *Messages* window. For extra guidance, consult the *"Messages Window"* help topic in PIPE-FLO's documentation, which can be accessed under Help ❯ Contents.

Level 0	
Level 1	
Level 2	
Level 3	
Level 4	

End of Tutorial

Tutorial #2:
Fluid Zones & Pipe Specifications

Tutorial #2 will consider the following concepts:

- Defining the properties of fluids to be simulated in a project
- Defining the materials and sizes of pipes to be used

In the first section of this tutorial, we will learn to specify the fluids (the *Fluid Zones*) which we wish to simulate in the FLO-SHEET. In the second section, we will learn to specify the pipes (the pipe *Specifications*) which we will need to create our piping system. To begin, create a new project (as discussed in **Tutorial #1**).

- Navigate to File » New; or
- Type Ctrl + N.

Again, it is recommended that you create your new project based on the default document outlined in **§1.1 Customizing** PIPE-FLO, and as guided in **Tutorial #1, Problem #1**. If you have not completed this tutorial yet, *now is a good time to go back and finish it.* Customizing PIPE-FLO as shown will ensure that the methods and results described herein will also be similar to those seen on your computer.

2.1 Defining Fluid Zones

With the new project created, the *"Fluid Zones"* dialog box can be accessed by clicking on its icon in the ribbon ( ), or by navigating to System » Fluid Zones. This can be seen in Figure 2.1. Subsequently, the *Fluid Zones* dialog box is pictured in Figure 2.2.

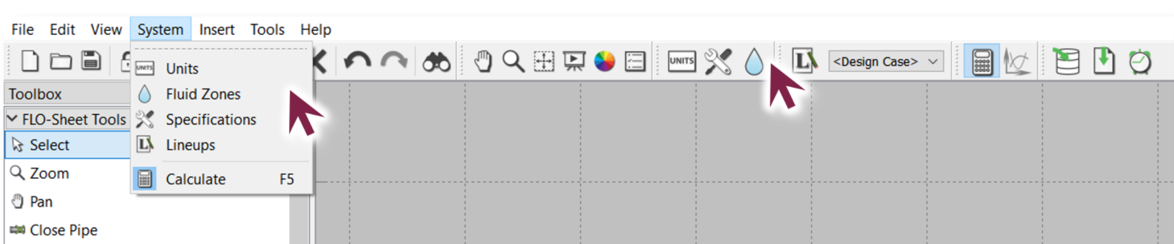


Figure 2.1: *Two methods of accessing the Fluid Zones dialog box.*

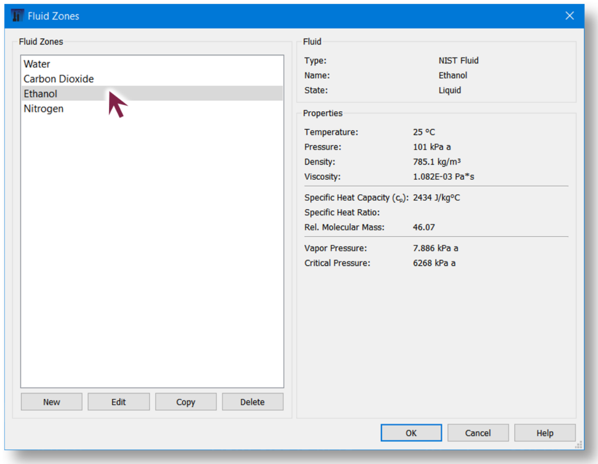


Figure 2.2: *The Fluid Zones dialog box, showing four "fluid zones" input by the user.*

To begin, click on the New button to create a new zone. This will allow you to define a fluid which will be present in your simulated piping system. The *"Fluid Properties"* dialog box will then open, as seen in Figure 2.3. Type a *unique* name for the fluid which you will be defining in the *"Fluid Zone Name"* field. The "property data" for your fluid can then be loaded. Under the *"Fluid Type"* drop-down menu, there are several possible fluid property packages.

- *<Undefined>:* no property package source has been chosen yet.
- *NIST:* the *National Institute of Standards and Technology* provides a (rather thorough) list of fluids which may be simulated. Data is obtained from cited literature sources, and correlations are used to model the fluids over a wide range of conditions.
- *Table:* the fluid properties of the three most commonly-used fluids—air, steam and water—as well as many other fluids are tabulated over a range of conditions. The data is also obtained from cited literature sources.
- *Custom:* allows the user to define their own fluids at specified conditions.
- *Stock:* the fluid properties of less common fluids, such as wood pulp slurries, are contained, here. This set of

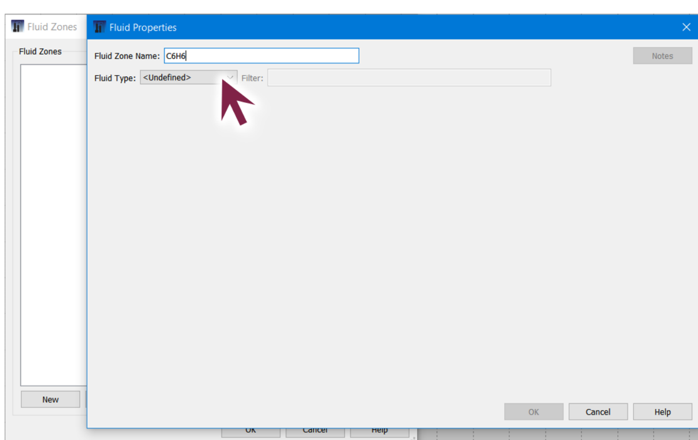


Figure 2.3: *The Fluid Properties Dialog box.*

data tends to be applicable in niche industrial applications and in cases where the fluids are complex. The data may be obtained from academic and/or industrial sources.

- *Custom Stock:* allows the user to define their own complex fluids (*e.g.* multiphase mixtures) at specified conditions.

Using NIST Data. The *NIST* data sets contain a wealth of information on a wide range of pure and complex fluids (*i.e.* mixtures). The data used in this feature is imported into Pipe-Flo from the free-to-use NIST **Re**ference **F**luid Thermodynamic and Transport **Prop**erties Database (*REFPROP*). Figure 2.4 illustrates the *Fluid Properties* dialog box, with the *NIST Fluid Type* option selected.

- The *"Available Fluids"* pane features a scrolling list of the fluids contained within the *NIST* dataset. Select one of the fluids to see its properties. In the case of the example in Figure 2.4, benzene has been selected.
- The *"Fluid"* pane shows the state in which the fluid exists at the chosen temperature and pressure. It also lists thermodynamic parameters such as the acentric factor; the meaning of these parameters will be covered in detail in Chemical Engineering 3D04 (Thermodynamics).
- The *"Fluid Properties"* pane requires the user to input an estimate for the temperature and pressure of the fluid in order to obtain fluid property data (*e.g.* viscosity, density, etc.). Be sure to input these values in the *units specified.* In particular, be sure to enter your pressure as the appropriate *absolute/gauge value.* As pressure (and sometimes temperature) will almost always change over the course of a piping system, these values tend to serve more as an initial estimate for Pipe-Flo and to determine the *state of matter* that the fluid occupies.

Unless heat transfer devices are incorporated into the Flo-Sheet, Pipe-Flo assumes *isothermal behaviour*—the temperature of the system does not change. If there are drastic temperature changes in your system, it is a good diea to employ multiple fluid zones. Two recommendations may be useful with regards to the specified pressure: (1) Input an "average" pressure of the fluid across the system as long as there is no phase change; or, (2) use an initial pressure that you know (*e.g.* the pressure of the fluid in a tank/inlet/outlet/etc.). Generally, your choice of temperature and pressure will not be highly consequential, as long as there are no phase changes and the input values are relatively close to the calculated values. However, changes in pressure and temperature influence gases more, so be cautious while using gases.

- The *Range of Applicability* pane provides the user with the minimum/maximum temperature and pressure values over which the property model is valid. Temperatures or pressures that fall outside this range will produce inaccurate results.
- The *Messages/Warnings* pane is used to provide the user with additional information regarding the fluid property model used. For example, a warning will be given if the temperature/pressure combination falls outside the range in which the model is applicable, or if the critical temperature and pressure are exceeded. Above the critical temperature and pressure, the fluid will exist as a *supercritical fluid*, having properties of *both liquids and gases.*
- The *Literature Reference* pane lists the source(s) of the correlations that Pipe-Flo uses to determine fluid properties.
- The Notes button opens a new dialog box, allowing the user to input comments regarding that particular fluid zone.
- The Convert to Custom button takes the property data from the current fluid and exports it as a "custom fluid". This allows the user to modify the property data and save these modified properties as a new fluid. See the upcoming *"Creating Custom Fluids"* section for more information.

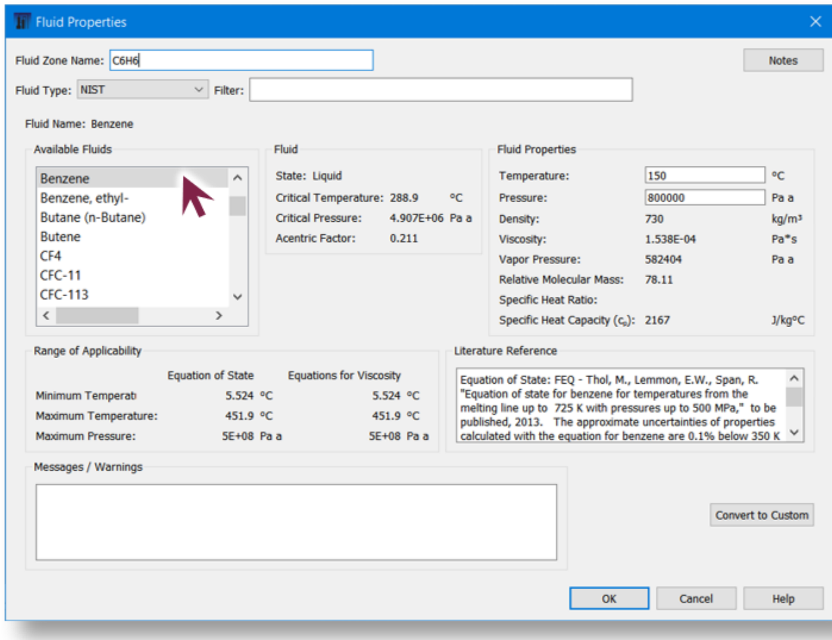


Figure 2.4: *The Fluid Properties Dialog box, using NIST property data to specify benzene as a fluid.*

At this point, try defining the fluid zone for benzene shown in Figure 2.4!

Using Data Tables. While both *NIST* and *Table* datasets contain rather obscure fluids, you are able still able to find common fluids like air, water and steam. Using the *Table* dataset to do this is show below in Figure 2.5. Like with the *NIST* data sets, select one of the available fluids, and enter the temperature and pressure; the *Fluid*, *Fluid Properties*, *Range of Applicability*, *Literature Reference*, and *Range of Applicability* panes are then used, as explained previously.

At this point, try defining the steam fluid zone shown in Figure 2.5!

Creating Custom Fluids. In some cases, PIPE-FLO will not have data available to model a particular fluid (*e.g.* carbon tetrachloride, nitric oxide, and mixtures such as kerosene, beer, etc.). The user is able to input thermodynamic data (under the "*Custom*" heading) so that PIPE-FLO can model these fluids. For the fluid to be modeled correctly, the user must input the following data:

- The fluid state of matter and critical pressure (optional for gases) under the "*Fluid*" pane.
- The fluid temperature, pressure, density, viscosity and vapor pressure (optional for gases) under the "*Fluid Properties*" pane.
- The relative molecular mass and specific heat ratio are mandatory for gases, but optional for liquids.
- The specific heat capacity is only required for heat transfer calculations.

Figure 2.6 illustrates the specification of carbon tetrachloride as a custom fluid. When specifying values, *be sure to check the units of pressure, and in particular, the pressure reference*!

At this point, try defining the carbon tetrachloride fluid zone shown in Figure 2.6!

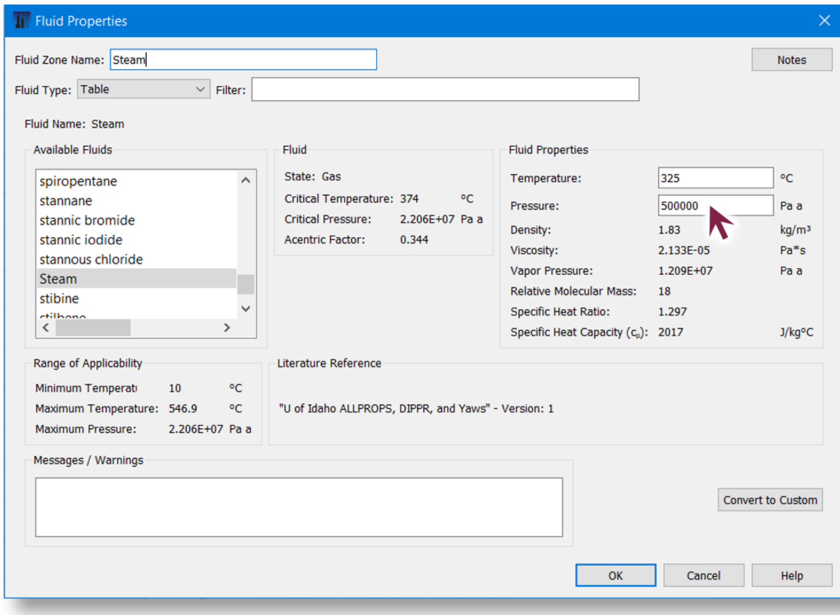


Figure 2.5: *The Table feature of the Fluid Properties dialog box, defining the properties of steam at 325°C and 500 kPa (absolute).*

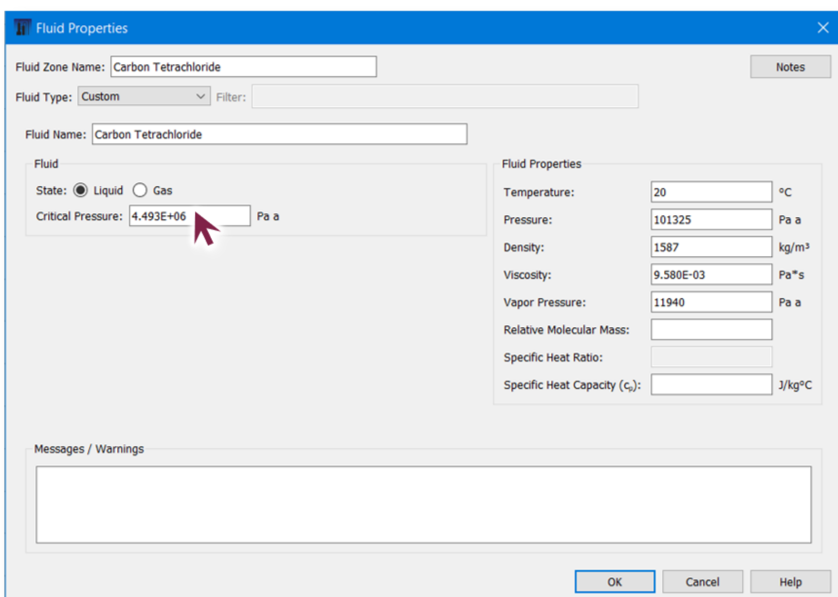


Figure 2.6: *Specifying carbon tetrachloride as a custom fluid. Note that properties required to fully define this fluid can be readily found on the internet. In this example, all values were sourced from Wikipedia's data page on carbon tetrachloride.*

Stock and Custom Stock Fluids. These features will not be covered in this tutorial, but operate much like the *Table* and *Custom* fluids.

Managing Fluid Zones. The *Fluid Zones* dialog box also contains a few options to manage your fluid zones that you have already created. Select a fluid zone, and choose an option:

- The Edit button allows the user to change the properties of an existing fluid zone. Perhaps, the fluid temperature in a zone has changed and it must be updated.
- The Copy button allows the user to duplicate an existing fluid zone. Perhaps, the user has already defined "water" at 50°C, but wishes to experiment with the system using water at 80°C.
- The Delete button allows the user to remove a fluid zone that is not in use.

Once the user has finished creating their fluid zone, they may select OK to continue working on their Flo-Sheet.

Creating or selecting a fluid zone will display the intrinsic properties of that fluid as a function of the fluid's temperature and pressure. Students can use the Pipe-Flo fluids databases—accessed using the *Fluid Zones* dialog box—to determine fluid properties for assignments and projects. For example, you may want to calculate the Reynold's number (Re) for water at 4°C and 101.3 kPa, given:

$$Re = \frac{\rho v D}{\mu} \tag{2.1}$$

Create a fluid zone at these conditions to easily determine the values of μ and ρ; these values can be read directly from within the *Fluid Properties* dialog box, as seen in Figure 2.7.

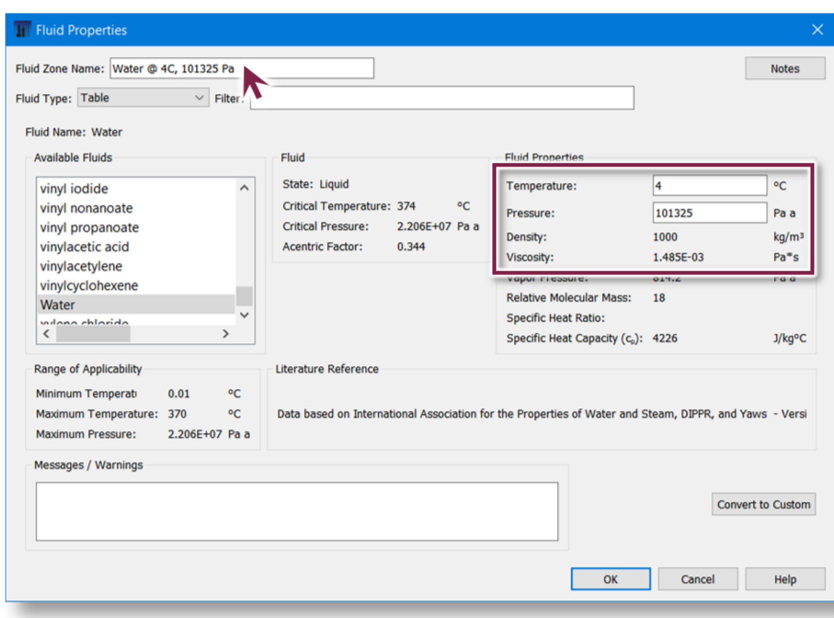


Figure 2.7: *Using* Pipe-Flo *to determine the properties of water at 4°C and 101325 Pa.*

2.2 Defining Pipe Specifications

We will now consider the creation of *pipe specifications*. This refers to choosing the material, diameter, and *schedule*[1] of the pipes that you wish to include in your Flo-Sheet. The *"Specifications"* dialog box is accessed by clicking on its icon in the ribbon ( ), or by navigating to System » Specifications, which can be seen in Figure 2.8. Subsequently, the *Specifications* dialog box is shown in Figure 2.9.

The *Specifications* dialog box operates very similarly to the *Fluid Zones* dialog box. Click the New button to create a new piping specification. The *"Pipe Properties"* dialog box will be opened, as seen in Figure 2.10. Like with *Fluid Zones*, type a unique name for the pipe.
To specify a pipe:

- Select a pipe material from the *Installed Pipe Tables* list on the left side of the dialog box; then
- Use the "❯" icon to expand the list of pipe schedules. Select a schedule from the list.

At this point, you have provided enough information in order to use this type of pipe in your Flo-Sheet. Click OK to proceed. Or, you may specify additional optional features.

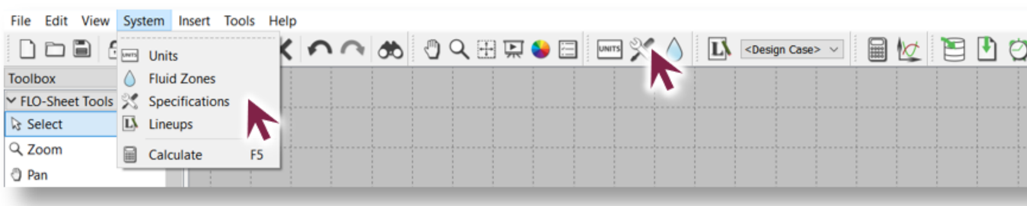


Figure 2.8: *Two methods of accessing the Specifications dialog box.*

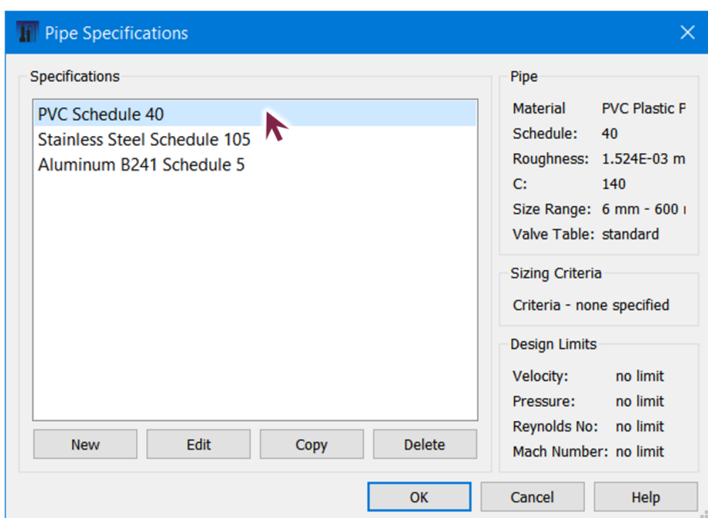


Figure 2.9: *The Specifications dialog box, showing three different pipe materials/schedules input by the user.*

[1]The pipe *schedule* is a part of a system which is used to describe the measurements of commercial pipes. For a given *nominal pipe size* (NPS), increasing the pipe schedule increases the wall thickness, while the actual outer pipe diameter remains constant. The schedule is generally related to a pipe's strength, however the nominal pipe diameter is not exactly equal to the pipe's actual outer diameter.

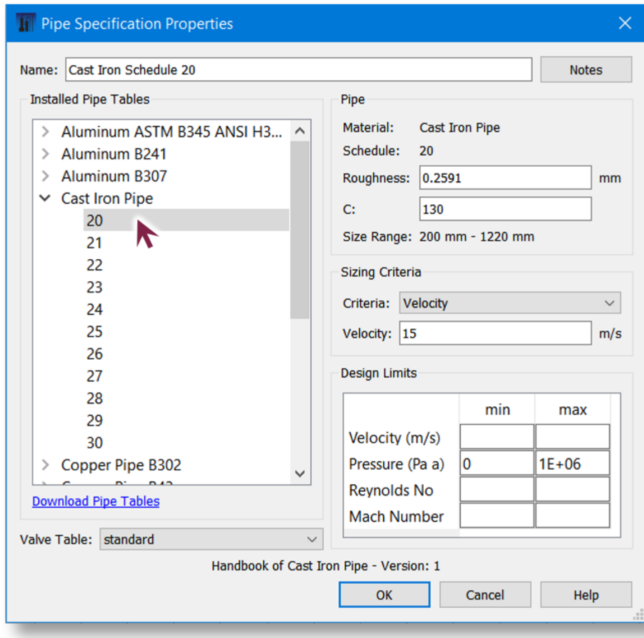


Figure 2.10: *The Pipe Properties dialog box with a specification for Schedule 20 cast iron pipe.*

- Under the *Pipe* pane, the absolute pipe roughness (e) and the Hazen-Williams roughness coefficient (C) can be found. These values may be modified for the pipe specified should they have more accurate or pertinent information. The "size range" refers to the range of nominal outer pipe diameters that are available with this pipe material/schedule combination.
- Under the *Sizing Criteria* pane, the user may choose a fluid design velocity, head loss per 100 units (m, ft, etc.) *or* pressure drop per 100 units, and PIPE-FLO will automatically determine the pipe diameter that most closely achieves this design criteria.
- Under the *Design Limits* pane, the user can set minimum and maximum limits on the fluid velocity, pressure, and Reynold's Number. If these set limits are exceeded during a simulation, a warning (Level 3) will be displayed in the *Messages* window. This a particularly useful feature as real-world pipes also have pressure and velocity limits relating to their materials and construction.
- The user can sometimes choose the table of valves that will apply to a particular pipe specification by choosing a "*Valve Table*" from the drop-down menu at the bottom-left of the *Pipe Properties* dialog box. Most often, the "standard" table is the only option available and will be sufficient for most applications.

At this point, try defining the Schedule 20 cast iron piping shown in Figure 2.10!

Like with *Fluid Zones*, the data in the *Pipe Properties* dialog box is referenced from the literature. The source of the data (often ASTM standards) can be found at the bottom of the dialog box.

Additional Pipe Tables. While PIPE-FLO does contain many different piping materials and schedules by default, there are other types of pipes available for download and installation by the user. Clicking the "`Download Pipe Tables`" hyperlink will bring the user to the developer's website where additional pipe tables can be found, complete with installation instructions.

Like the *Fluid Zones* feature, students can use the PIPE-FLO pipe databases—accessed using the *Specifications* dialog box—to determine the properties of pipes for assignments and projects. For example, the pipe roughness (e) or Hazen-Williams roughness coefficient (C) can be ascertained. In this course, you will be asked to calculate the friction factor for a given pipe (made of a given material and of a given diameter) using the Haaland equation:

$$\textbf{Haaland Equation} \qquad \frac{1}{\sqrt{f}} = -1.8 \log\left[\left(\frac{e/D}{3.7}\right)^{1.11} + \frac{6.9}{Re}\right] \tag{2.2}$$

Create a pipe of the given material to easily determine the value of e; this value can be read directly from the *Pipe Properties* dialog box, as seen in Figure 2.10.

Tutorial Problems

Problem #1. If you have not already done so, complete the following:

1. Create the benzene, steam, and carbon tetrachloride fluid zones defined in Figures 2.4, 2.5, and 2.6, respectively.
2. Create the Schedule 20 cast iron piping specification shown in Figure 2.10.

Recall that in this courseware, *all pressures are expressed in absolute terms*!

Problem #2. Complete the following exercises:

(A) Fill in the table below by using PIPE-FLO's fluid properties databases.

Fluid	T (°C)	P (kPa a)	State	ρ (kg/m^3)	μ (Pa·s)
R-600 refrigerant	-20	150	Liquid	621.8	2.506×10^{-4}
Benzene	150	800			
Steam	325	500			
Octane	250	350			
CFC-115 refrigerant	-80	1300			

(B) Fill in the table below by using PIPE-FLO's pipe specifications databases.

(C) For a given pipe material, what happens to the pipe roughness when the pipe schedule is varied?

Material	Schedule	Size Range (mm)	e (mm)	C (−)
Aluminum B241	60	200–300	1.524×10^{-3}	140
Cast Iron	20			
PVC Plastic Pipe	80			
Copper Pipe B42	STD			
Stainless Steel ASME B36.19M	10S			

Problem #3. With benzene as a model fluid, elucidate the following trends using a spreadsheet tool. Use a secondary axis to plot both density and viscosity on the same graph, and use a new graph for each question.

(A) What effect does *pressure* play on the viscosity and density of a *liquid*? At a constant temperature of $T = 20°\text{C}$, plot μ and ρ as a function of P over an order of magnitude, from 100 kPa to 1,000 kPa.

(B) What effect does *pressure* play on the viscosity and density of a *gas*? At a constant temperature of $T = 200°\text{C}$, plot μ and ρ as a function of P from 100 kPa to 1,000 kPa. Use a secondary set of axis and plot both trends.

(C) What effect does *temperature* play on the viscosity and density of a *liquid*? At a constant pressure of $P = 2,000$ kPa, and plot μ and ρ as a function of T from 20°C to 200°C.

(D) What effect does *temperature* play on the viscosity and density of a *gas*? At a constant pressure of $P = 0.1$ kPa, and plot μ and ρ as a function of T from 20°C to 200°C.

(E) Summarize your (important) findings in the table, below, for *both* liquids and gases:

The effect of ❯ on ❯.	**Pressure, P**	**Temperature, T**
Viscosity, μ		
Density, ρ		

The results of this question comprise a *key course takeaway*: for a liquid, density and viscosity are essentially independent of pressure. Even though pressure changes across your piping network, it is not likely to substantially affect your values of ρ and μ. Therefore your choice of pressure value when creating a fluid zone is not particularly crucial for liquids. *However*, gases are a completely different story!

End of Tutorial

Tutorial #3:
Simple Fluid Statics

Tutorial #3 will consider the following concepts:

- Using fluid zones and pipe specifications in the FLO-SHEET
- Adding tanks and pipes to the FLO-SHEET
- Specifying tank and pipe parameters
- Running and interpreting simulations

Having completed the first two tutorials, you are now ready to create your first PIPE-FLO FLO-SHEET! A simple example of pressure-driven flow between two tanks will be introduced. As illustrated in Figure 3.1, a system with a water tower (feed reservoir) and a receiving basin will be modeled. In this example, both tanks are open to the atmosphere and sit on level ground. Therefore, the inlet/outlet pipes are at the same elevation ($\Delta z = 0$). However, the water level in the tower and basin are 100 m and 1 m, respectively. The tanks are separated by a 100 m long, 200 mm (nominal) diameter Schedule 20 cast iron pipeline at ground-level through which water at 20°C flows. As PIPE-FLO can only simulate *steady-state systems*, we will assume that the *water tower and receiving basin are fed at the same rate that they are drained*. We will investigate this scenario in the following tutorial.

Work along with the text in order to get the most out of this tutorial.

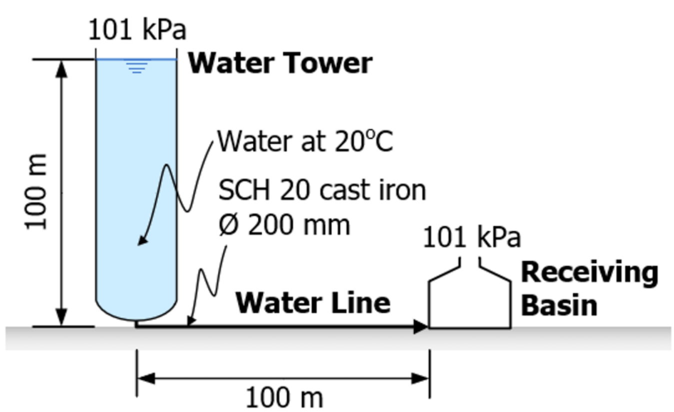


Figure 3.1: *A water tower and receiving basin connected by a pipe.*

⚠ Are you using the *default document* created in Tutorial #1? Ensure that the *Laminar Cutoff Re* is set to 2300 to ensure consistent results! Also, here is one final reminder that *pressures values always refer to absolute pressures* in this courseware.

3.1 Preparing Fluid Zones and Pipe Specifications

We will begin by defining the fluid zones and pipe specifications that we will need in order to construct the FLO-SHEET. As we have already discussed these topics in detail, we will not focus on them as closely, here. Should you require extra assistance, refer back to **§2.1 Defining Fluid Zones** and **§2.2 Defining Pipe Specifications**.

Fluid Zones. Start by creating a "water" fluid zone, where:

- $T = 20°\text{C}$
- $P = 101$ kPa.

Pipe Specifications. Next, create a "Schedule 20 cast iron" pipe specification. The *Sizing Criteria* and *Design Limits* features will not be used in this FLO-SHEET. These two actions are shown in Figure 3.2.

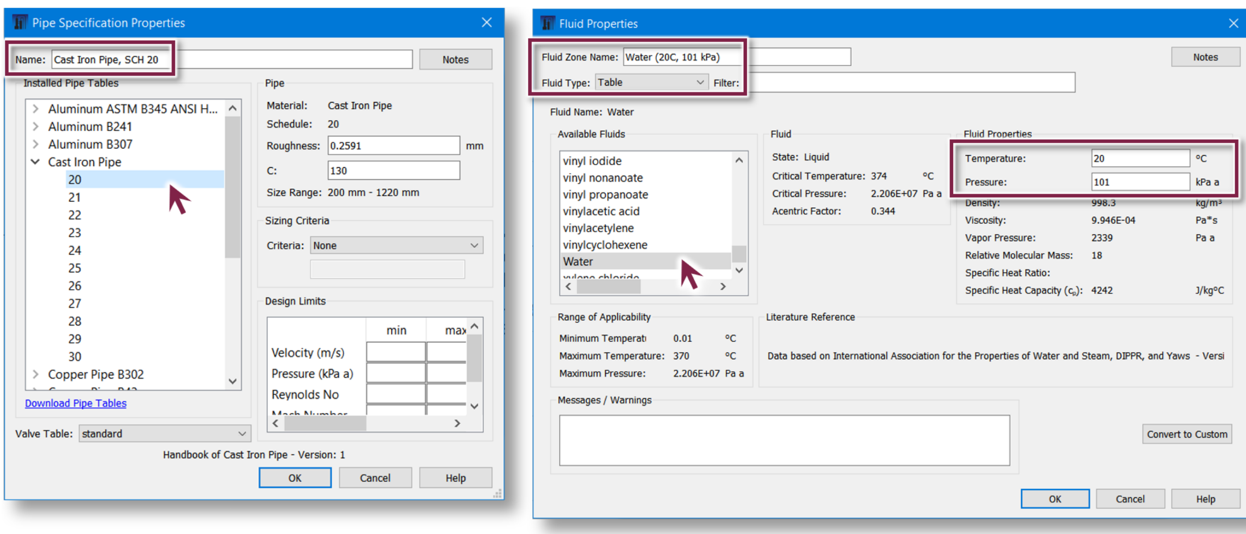


Figure 3.2: ***(L)*** *The "water" fluid zone at 20°C and 101 kPa, and* ***(R)*** *the Schedule 20 cast iron pipe specification.*

3.2 Preparing Tanks and Pipes

Tanks. We will continue by adding our two tanks to the FLO-SHEET. This can be done by selecting the *Tank* device from the *Toolbox* (under *Basic Devices*), or by navigating to Insert › Basic Devices › Tank, as seen in Figure 3.3. Subsequently, click on the locations in your flowsheet where you would like your tanks to appear. The black circle on the "tank" cursor marks where the center of the tank will be.

ℹ The location of a piece of equipment (*e.g.* tanks, pumps, etc.) on the FLO-SHEET does not have a bearing on its operation. For example, placing one tank above another on the FLO-SHEET does not give PIPE-FLO any information above the elevation difference in the tanks, or the distance between them. It is *good form* to place equipment at the same elevation on the same horizontal gridline, or if a tank is located on a hill above another tank, place the first tank at a "higher" location on the FLO-SHEET.

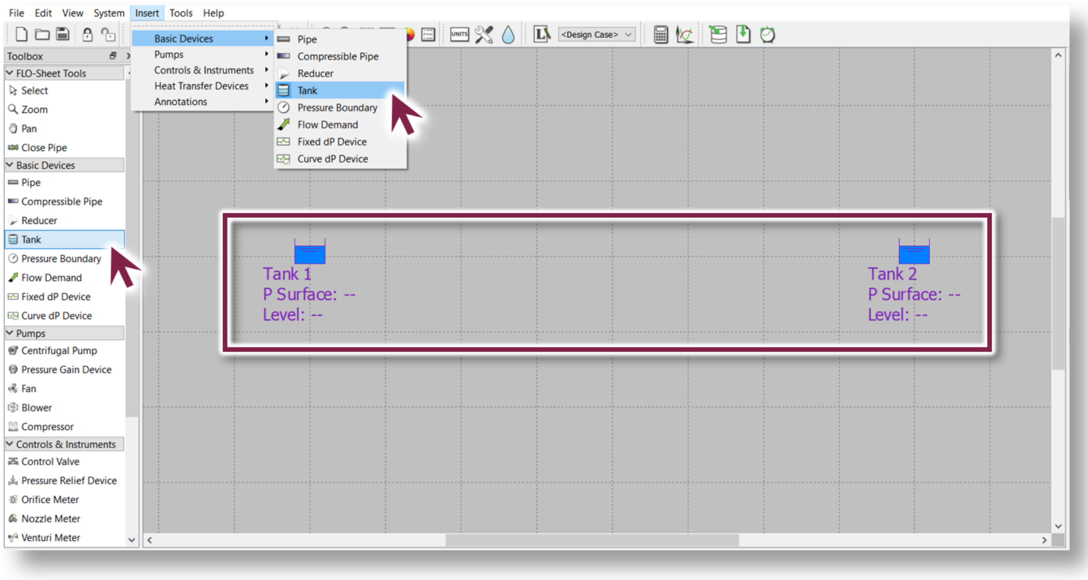


Figure 3.3: *Two ways of adding Tank devices to the* FLO-SHEET.

Pipes. Now, we will add a pipe that joins the two tanks, as seen in Figure 3.4. From the *Toolbox* window under *Basic Devices*, select the *Pipe* icon; or, navigate to Insert » Basic Devices » Pipe. To connect the two tanks:

1. Click on the first tank to begin the pipe there.
2. Move your mouse to the second tank; the pipe will extend from the first tank to your mouse cursor as you move it. Click the second tank to attach the end of the pipe there.

If you hovered your mouse over the first tank before the pipe was connected, you would notice that four green arrows would appear around the tank, as seen in Figure 3.5. This means that you would be able to start your pipe in this location. If the arrows were red, PIPE-FLO would not allow you start a pipe in that location. For example, this would occur if you already had a pipe *leaving* this tank.

Notice the arrowhead at the right terminus of the pipe. PIPE-FLO assumes that when you are using the *Pipe* tool, the *first* location that you click refers to the *upstream end* of the pipe and the second location that you click refers to the *downstream end* of the pipe. Thus, the arrow on the pipe *points in the intended direction of flow.* During a simulation, if the flow in the pipe occurs in the opposite direction from what is intended, PIPE-FLO will indicate a *Level 3* "Reversed Flow" error in the *Messages* window. For this tutorial, make sure that the direction of flow is from *Tank 1* towards *Tank 2*.

Specifying System Parameters. It is now time to assign tank and pipe parameters. Recall from the preamble at then beginning of the tutorial, and illustrated in Figure 3.1:

- Both tanks sit on flat level ground.
- The pipe exits the water tower at ground level, continues horizontally, and enters the receiving basin also at ground level.
- The height of water in the tower (h) is 100 m, while the receiving basin contains 1 m of liquid.
- Both tanks are open to the atmosphere; $P_{atm} = 101$ kPa.

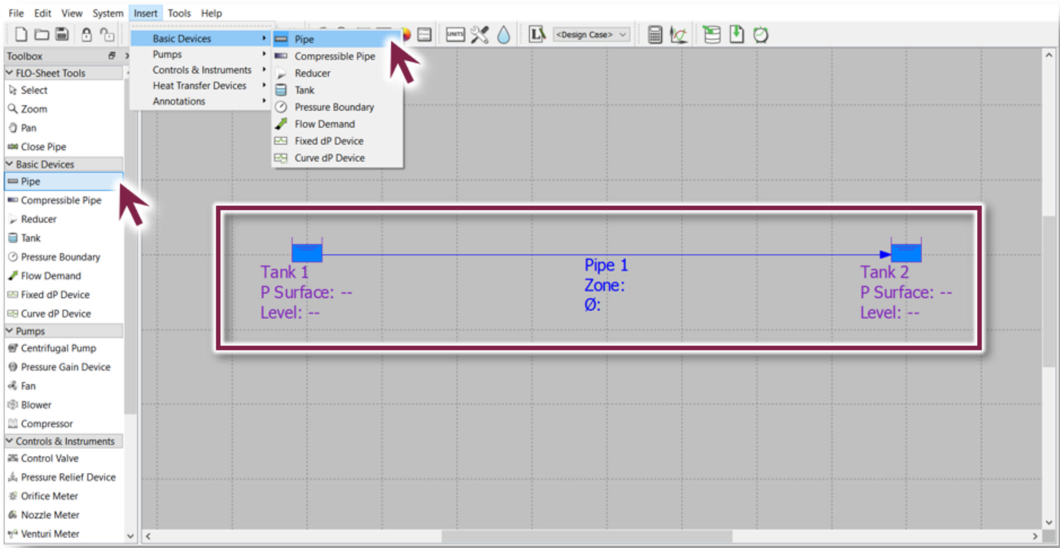


Figure 3.4: *Two ways of adding a pipe to the* FLO-SHEET *to connect the two tanks.*

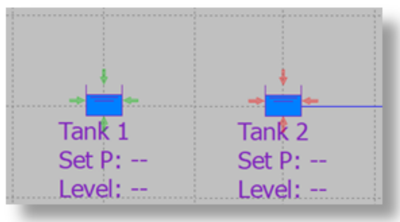


Figure 3.5: *The four green arrows indicates that the user can connect the start or end a of pipe in this location. Here, the red arrows indicate that the user cannot begin a pipe at Tank 2 as a pipe has already be started there.*

Choose the *Select* tool from the *Toolbox*, or press Esc to cancel any previously-engaged tool. Click on the *upstream tank*. This will immediately display the "*Tank Design*" heading in the *Property Grid*. Type the following information into the fields under *Tank Design*, as seen in Figure 3.6:

- Choose a *Name* for your tank. Here, we have selected "Water Tower".
- PIPE-FLO automatically changes the *Design Status* from *Not Designed*, to *Sizing*, to *Designed*, when sufficient operational and design data have been entered, respectfully. A device will automatically turn from purple to black when the *Design Status* changes from *Not Designed* to *Sizing*, indicating that device has enough information for calculations.
- The *Elevation* can be set to zero.
- The tank surface pressure is simply the atmospheric pressure, $P_{atm} = 101$ kPa, as the tank is open to the air.
- The *Liquid Level* is 100 m as this is the height of water in the tower.
- The *Fluid Zone* field will read <*Undefined*> by default. This can be changed by clicking on the field and choosing your "Water" fluid zone from the drop-down menu.
- The *Geometry* field allows you to change the shape of your tank along with its corresponding dimensions. This includes spherical, rectangular, cylindrical or custom shapes. It is important to note that changing the geometry of a tank *will not have any affect on the system.* Calculations performed using the Bernoulli Equation will

produce results (e.g. flow rates, pressures, etc.) *independent of the tank geometries.* For all of these tutorials, we will leave the geometry as <*Unspecified*>.

- Enter the location of pipe *Penetrations* into the tank. A *Penetration* is the height above the tank bottom *i.e.* above the *elevation* where a pipe enters or leaves the tank. As our pipe named *Pipe 1* leaves the water tower at its very bottom, the *Penetration* height is 0 m—the default value which is already entered. By clicking on the *Penetrations* field, and entering a new "*Height*" in the *Tank Penetrations* dialog box, you could change this value if you desired to do so, as shown in Figure 3.6.

> Tank penetrations are assumed to be at the bottom of the tanks (0 m) in subsequent tutorials, unless otherwise specified.

Now, apply the *Tank 2* specifications which are similar to those of the water tower, as seen in Figure 3.7.

- Choose a new *Name* for this tank. Here, the "Receiving Basin" has been selected.
- The *Liquid Level* is 1 m, as prescribed.

> You may wonder why an elevation of $z = 0$ m was selected for the water tower or the receiving basin. It was specified that the two tanks are on level ground, thus their elevations are identical. Consider the Bernoulli Equation (with no frictional losses) which PIPE-FLO uses to calculate flow conditions such as the velocity and the volumetric flow rate:
>
> $$\frac{P_1}{\rho} + \alpha_1 \frac{v_1^2}{2} + g z_1 = \frac{P_2}{\rho} + \alpha_2 \frac{v_2^2}{2} + g z_2 \tag{3.1}$$

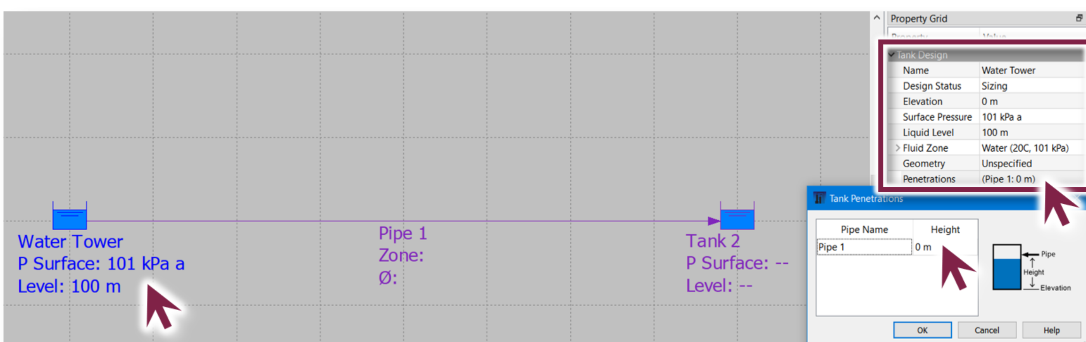


Figure 3.6: *Specifying the design parameters of the water tower.*

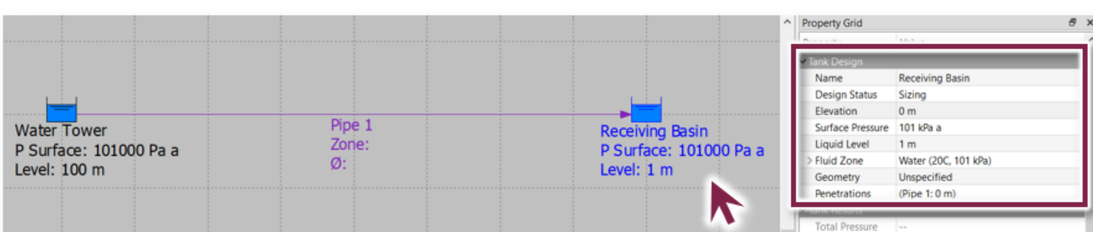


Figure 3.7: *Specifying the design parameters of the receiving basin.*

By inspection, the potential energy terms gz_1 and gz_2 clearly cancel each other out when $z_1 \equiv z_2$ for *any choice of* z_1, z_2. Thus, the choice of elevation is arbitrary in this scenario, as long as both tanks are given the same elevation value.

Finally, we will input the required data for the pipe. Again, using the *Select* tool, click on the pipe and the "*Pipe Design*" heading will appear in the *Property Grid*. Type the following information into the fields under *Pipe Design*, as seen in Figure 3.8:

- Choose a *Name* for your pipe. Here, "Water Line" was selected.
- The *Fluid Zone* field will read <*Undefined*> here, as well. Click on the field and choose your "Water" fluid zone from the drop-down menu.
- The *Specification* field will read <*Undefined*> by default. Click on the field and choose your "Cast Iron Pipe" specification from the drop-down menu.
- Enter the *Size* of the pipe that you wish to specify. PIPE-FLO is looking for the *nominal pipe diameter*; in our case, this value is 200 mm. See below for a more detailed explanation of the "nominal pipe diameter".
- Type the total length of the pipe into the *length* field; here, the value is 100 m.
- The *K (Valves & Fittings)* field is used to specify the minor loss coefficients associated with valves and fittings in this particular segment of pipe. In this example, we will neglect the effects of valves and fittings; the value of this field will be left as "--".
- The *Close Pipe* checkbox simulates the effect of closing off this particular pipe using a valve when this box is checked. In our example, we want there to be flow through the pipe, so leave the box unchecked.
- When checked, the *Prevent Backflow* checkbox simulates the effect of installing a check valve on the pipe in order to prevent fluid from flowing in the wrong direction. Interesting results may be obtained in this example when there is no prevention of backflow, so we will leave this box unchecked.

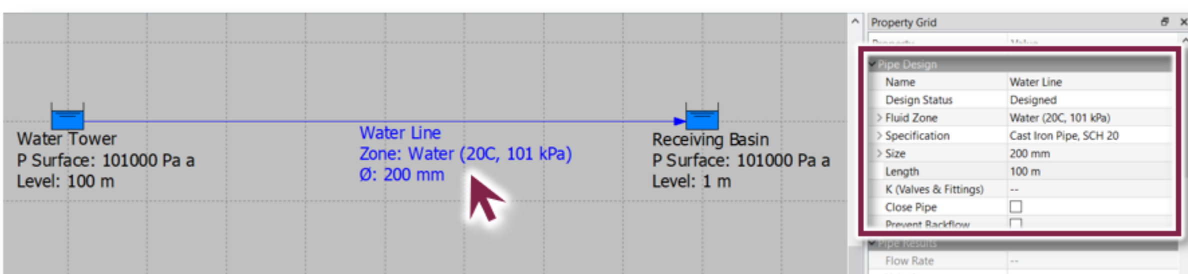


Figure 3.8: *Specifying the design parameters of the water line.*

As mentioned previously, PIPE-FLO asks for a *nominal pipe diameter* when specifying the *Size* of the pipe. However, this *does not mean* that the fluid flows through a channel the size of the nominal diameter. In fact, the *actual hydraulic diameter* of the pipe—the one that you would use in fluid mechanics calculations (*e.g.* finding the Reynold's number, using the Bernoulli Equation, etc.)—is referred to in industrial circles as the *internal diameter* (I.D.). The nominal pipe diameter is more closely related to the *outer diameter* (O.D.) of the pipe: the nominal diameter is a "rough rounding" of the value of the O.D. The relationship between the nominal pipe diameter and pipe O.D. can easily be found online. As the nominal diameter is not useful in fluid mechanics calculations, PIPE-FLO can display the actual I.D. of the pipe after the nominal diameter has been input. Click the "❯" next to the *Size* field to view the pipe I.D., as seen in Figure 3.9. For the SCH 20 cast iron pipe with a nominal diameter of 200 mm, the I.D. is 212.1 mm.

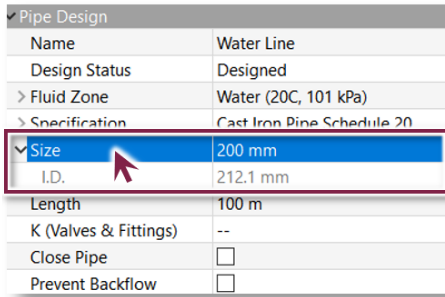


Figure 3.9: *Determining a pipe's internal diameter.*

We are now ready to begin our first simulation!

⚠ Remember to save your work: press Ctrl + S, navigate to File ❯ Save, or click the *Save* icon on the ribbon.

3.3 Fluid Statics Simulations

It is often the case that students build their FLO-SHEET first, then begin their simulation once the entire FLO-SHEET has been completed—like we have done with this tutorial. While this approach is valid, it is not necessarily the best way to proceed. It is also possible to build your FLO-SHEET *while* in *Calculation Mode*, an approach which can have several advantages:

- This is a *parallel* approach, which tends to be more efficient than *serially* building your FLO-SHEET, then performing a simulation.
- Perhaps more importantly, leaving *Calculation Mode* "on" allows PIPE-FLO to give you feedback on your FLO-SHEET in real time. Suppose that you forget to include a particular piece of data in the *Tank Design* field. PIPE-FLO will alert you with a warning in the *Messages* window.

As a result, it is "good practice" to run your simulation while designing the FLO-SHEET. It is much easier to address problems one-at-a-time than all-at-once. Keep this in mind for future tutorials and for your own work.

Recall from **§1.2 Navigating the PIPE-FLO Interface** that *Calculation Mode* can be entered toggling on the *Enter/Exit Calculation Mode* icon on the ribbon. *Calculation Mode* may also be toggled on/off by navigating to System ❯ Calculate, or by pressing F5. This is illustrated in Figure 3.10.

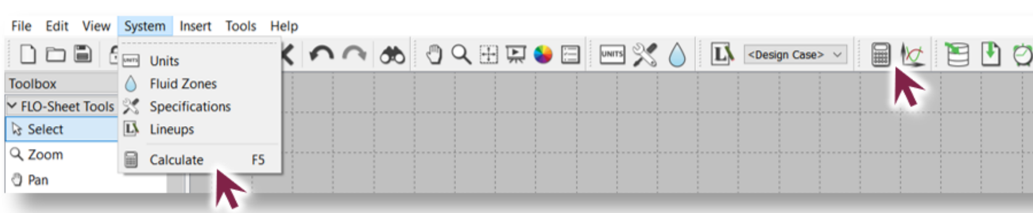


Figure 3.10: *Entering and exiting Calculation Mode.*

Begin your simulation by turning *Calculation Mode* on.

Immediately, you should see that your FLO-SHEET changes to display the results of the calculation; you may notice that some results are displayed, as seen in Figure 3.11. Furthermore, the *Messages* window indicates that the calculation is complete, the number of iterations, and the standard deviations associated with these iterations.

If incorrect results are shown and/or there are warnings in the *Messages* window, you may have erred in entering your data. Recheck your values against those used in this tutorial.

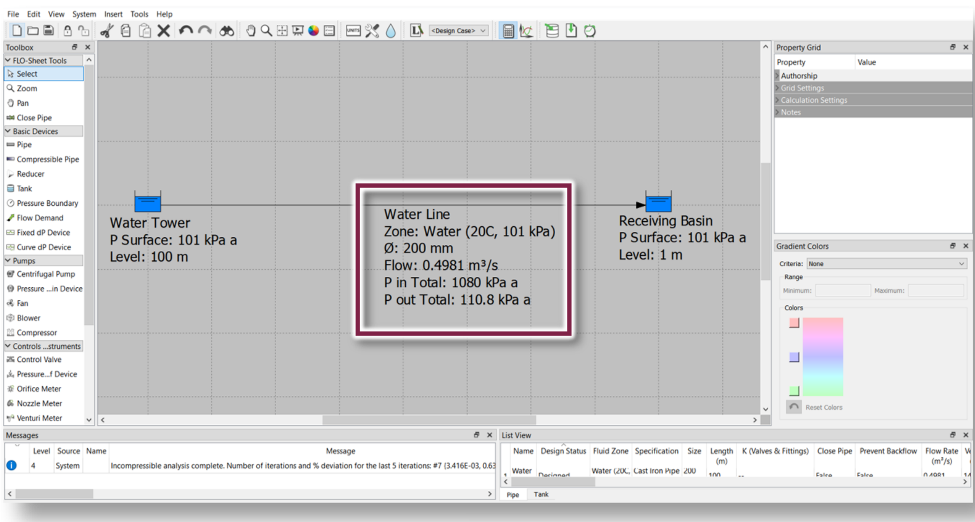


Figure 3.11: *The* FLO-SHEET *upon entering calculation mode. Particular system parameters and the results for the flow rate through the water line are directly displayed on the* FLO-SHEET.

You may wish to see more of the results, such as the fluid velocity in the water line, or other simulation parameters. There are several ways that this can be accomplished:

Visualizing Simulation Results on the FLO-SHEET. Simulation results can be displayed directly on the FLO-SHEET by changing the *Device View Options*, as mentioned in **§1.2 Navigating the PIPE-FLO Interface**. To enter the *Device View Options* dialog box, click the icon on the ribbon ( ), navigate to View » Device View Options, or press Ctrl + D. This can be seen in Figure 3.12. Inside the dialog box, the user can select the parameters that they would like to be displayed on the FLO-SHEET by scrolling through the list and checking boxes. To narrow down the selection of parameters, select one of the *Devices* from the left menu. The choice of parameters to display is personal, but for this tutorial, we will enable on the FLO-SHEET:

- For *Pipes*: "Name", "Size", "Length", "Flow Rate", "Velocity", and "Reynolds Number". Other useful parameters include: "Pressure Drop", "Head Loss", and "Pipe Friction Factor".
- For *Tanks*: "Name", "Elevation", "Surface Pressure", "Liquid Level", and "Calculated Total Pressure".
- All other parameters can be left as-is, for the time being.

These modifications are illustrated in Figure 3.13, while the results on the FLO-SHEET can be seen in Figure 3.14.

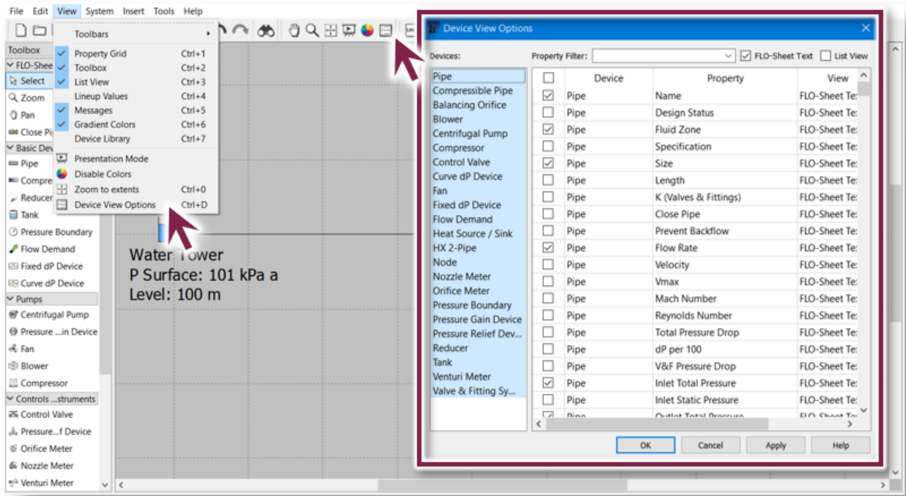


Figure 3.12: *The Device View Options dialog box.*

Try displaying these results on your FLO-SHEET, as shown!

Since you now know how to change the results displayed on your FLO-SHEET, note that in subsequent tutorials, you may have different information on your screen than in the figures shown. It is suggested that you change your displayed results to match those displayed in these tutorials.

PIPE-FLO uses a set of abbreviations for the results displayed on the FLO-SHEET. These correspond to the enabled properties in the *Device Display Options*. In the example in Figure 3.14, the abbreviations are interpreted as the following:

- ∅, *L*, *Flow*, *Vel*, and *Re* refer to the pipe's nominal diameter, length, (volumetric) flow rate, average velocity, and Reynolds Number, respectively.

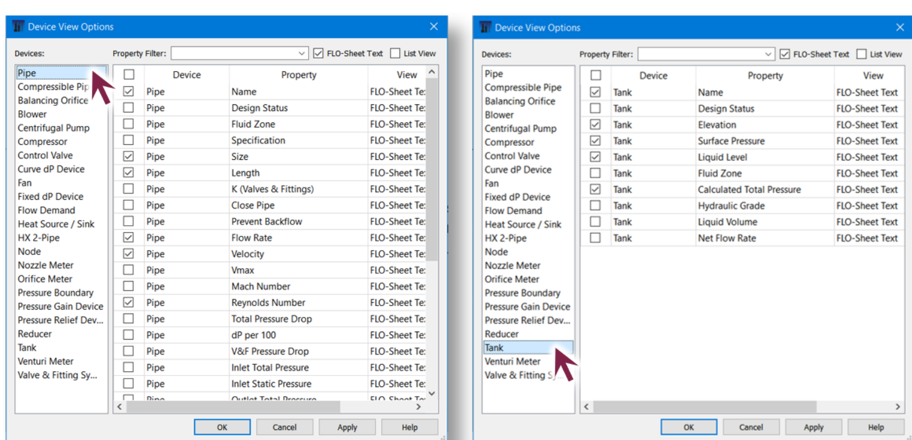


Figure 3.13: *Setting the Device View Options for pipes and tanks.*

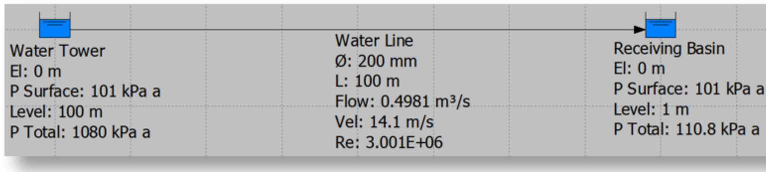


Figure 3.14: *Displaying simulation results directly on the* FLO-SHEET.

- *El*, *P Surface*, *Level*, and *P Total* refer to the tank elevation, surface pressure, fluid height (level), and pressure at the tank bottom, respectively.

The keen observer may realize that the *Set P*, *Level*, and *P* are inherently related. Redefining *Set P* as P_0, *Level* as h, and *P* as *Pressure*, we can use the basic equation of fluid statics to relate these three variables (and fluid density, ρ) to one another:

$$P = P_0 + \rho g h \tag{3.2}$$

We can use this equation to verify the results of the PIPE-FLO simulation using values from the water tower.[1] Given that:

- $P_0 = 101$ kPa or $101 \times 10^3\ \mathrm{kg \cdot m^{-1} \cdot s^{-2}}$ (as prescribed in problem);
- $\rho_{\text{water}} = 998.3\ \mathrm{kg \cdot m^{-3}}$ (from *Fluid Zones*);
- $g = 9.81\ \mathrm{m \cdot s^{-2}}$; and
- $h = 100$ m (as prescribed in the problem).

We will use these values to calculate the pressure at the bottom of the tank, P:

$$P = \left(101 \times 10^3\ \frac{\mathrm{kg}}{\mathrm{m \cdot s^2}}\right) + \left(998.3\ \frac{\mathrm{kg}}{\mathrm{m^3}}\right)\left(9.81\ \frac{\mathrm{m}}{\mathrm{s^2}}\right)\left(100\ \mathrm{m}\right)$$

$$\boxed{P = 1.08 \times 10^6\ \frac{\mathrm{kg}}{\mathrm{m \cdot s^2}} \quad \text{or} \quad P = 1.08 \times 10^6\ \mathrm{Pa}}$$

From Figure 3.14, PIPE-FLO found that the pressure at the bottom of the tank, P, is 1.08×10^6 Pa, matching the value from our calculation. The results are verified!

Let us perform one more verification before we move on—this time, on the water line. Recall that the formula to calculate the Reynolds Number in a circular pipe is as follows, where v is the average fluid velocity (v_{avg}), and D is the hydraulic diameter of the pipe:

$$Re = \frac{\rho v D}{\mu} \tag{3.3}$$

Given the following values:

[1] We will perform such verifications from time to time in order to deepen your understanding of key course concepts.

- $\rho_{\text{water}} = 998.3 \text{ kg} \cdot \text{m}^{-3}$ (from *Fluid Zones*);
- $v = 14.1 \text{ m} \cdot \text{s}^{-1}$ (from the simulation results);
- $D = 212.1$ mm, or 0.2121 m (the *inner* pipe diameter, retrieved from under the *Pipe Design* » *Size* heading in the *Property Grid*); and
- $\mu_{\text{water}} = 9.946 \times 10^{-4}$ Pa · s, or $9.946 \times 10^{-4}\ kg \cdot m^{-1} \cdot s^{-1}$ (from *Fluid Zones*).

Using these values, *Re* can be calculated, as follows:

$$Re = \frac{\left(998.3\ \frac{\text{kg}}{\text{m}^3}\right)\left(14.1\ \frac{\text{m}}{\text{s}}\right)\left(0.2121\ \text{m}\right)}{9.946 \times 10^{-4}\ \frac{\text{kg}}{\text{m}\cdot\text{s}}}$$

$$\boxed{Re = 3.002 \times 10^6}$$

From Figure 3.14, Pipe-Flo found that the Reynolds Number in the water line, *Re*, is 3.001×10^6, which is very close to the value from our calculation. The results are verified, again!

Comprehensive Simulation Results in the *Property Grid*. On a simple Flo-Sheet, directly displaying calculated results can be very useful, however, a more complex Flo-Sheet will become too crowded with all the properties displayed. In these cases, the user should disable the extraneous properties in the *Device View Options* and find all the results tabulated in the *Property Grid*. In fact, *comprehensive* simulation results for each object on the Flo-Sheet can been found in the *Property Grid* under the heading "*<Equipment> Results*", where *<Equipment>* refers to a functioning *Basic Device, Pump, Control* or *Instrument*, etc. The *<Equipment> Results* heading is found directly below the *<Equipment> Design* heading in the *Property Grid*. Displaying a particular simulation result directly on the Flo-Sheet generally requires the user to intervene in the *Device Display Options* dialog; however, all possible results are automatically displayed in the *Property Grid* without any additional action by the user.

Locate the results for the receiving basin and water pipe in the *Property Grid*!

At this point, save your work. We will use the Flo-Sheet that you have just created (both in the tutorial problems, and in the subsequent tutorial) and refer to it as the "base case" scenario.

Tutorial Problems

Problem #1. Using the "base case" scenario generated in this tutorial, probe the effects of changing the following parameters (individually) on our fluid statics system. How does the volumetric flow rate, average velocity, and Reynolds Number in the water line change? Revert back to the "base case" prior to starting a new part of the question. Use the summary table found below.

(A) Suppose the region is experiencing drought conditions and the fluid level in the water tower falls to 10 m. Change the *Liquid Level* in the water tower to "10 m" and report the new values of volumetric flow rate, average velocity, and Reynolds Number in the water line.

(B) Suppose that the receiving basin is not draining properly and its fluid level rises to 25 m. Change the *Liquid Level* in the receiving basin to "25 m" and report the new values.

(C) Suppose that receiving basin is uphill of the water tower: the bottom of the receiving basin is 40 m higher than the bottom of the water tank. Change the *Elevation* of the receiving basin to "40 m" and report the new values.

(D) Suppose that the distance between the water tower and receiving basin is increased: the length of the water line extends to 1 km. Change the *Length* of the water line to "1000 m" and report the new values.

(E) Suppose that this system is not for water, but for gasoline, instead. PIPE-FLO does not have "gasoline" as fluid installed by default, so we will use octane to model gasoline, here. Create an "octane" fluid zone at 15°C and 101 kPa and re-run the simulation, remembering to change the *Fluid Zone* in the two tanks and pipe. (You may also want to change the *Names* of the tanks and pipe!) What are the new values after the fluid is changed?

(F) Suppose that the water line must be rebuilt using Schedule 120 PVC plastic pipe with a nominal diameter of 100 mm for reasons of cost. Remember to change the *Specification* of the water line! What are the new values after the piping is changed?

Problem	**Volumetric Flow Rate (m^3/s)**	**Average Velocity (m/s)**	**Reynolds Number**
Base Case	0.4981	14.1	3.001×10^6
#1A			
#1B			
#1C			
#1D			
#1E			
#1F			

Problem #2. Starting with "base case" scenario generated in this tutorial, answer the following questions about redesigning the water supply system. The following questions are cumulative. *Do not go back to the base case with each new part.*

(A) Suppose that a new receiving basin is needed, however, it must be constructed in a new location further away from the water tank. What is the maximum allowable distance between the water tower and receiving basin while the volumetric flow rate of water remains above 0.30 m^3/s? Vary the water line's *Length*.

(B) After constructing the new receiving basin and water line at the maximum possible distance (from **Part (A)**), it appears that a slightly higher flow rate of water is required due to a civil engineering error. Your colleague suggests that increasing the pressure above the fluid in the water tank (using an inert gas) will increase the flow rate of the water entering the receiving basin. Find the approximate *Surface Pressure* of the water tank that will achieve a volumetric flow rate of 0.35 m^3/s. Vary the water tower's *Surface Pressure*.

(C) Your colleague suggests that due to the pressurization in the newly-built system, the water tower could be used to supply a receiving basin built *even higher* than the water level in the tower. (Albeit, the water flow rate would be lower.) See the diagram, below in Figure 3.15. Is your colleague correct in his theory? What is the maximum difference in elevation, z, between the two tanks? Vary the *Elevation* of the receiving basin. Keep the water line's *Length* at the value found in **Part (A)** and the and water tower's *Surface Pressure* at the value found in **Part (B)**.

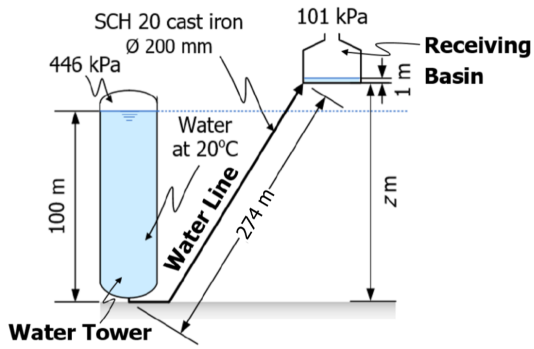


Figure 3.15: *Diagram for* ***Problem #2D.***

Problem #3. Denver, Colorado is known as the "Mile-High City" due to its elevation of one mile (1609 m) above sea-level. As is such, the air pressure in Denver is significantly less than the 101.325 kPa benchmark that we use in this courseware. Let us simulate the effect of moving our water distribution system to Denver. Again, the following question are cumulative, so begin with the "base case" and do not go back to it with each new part.

(A) What is the atmospheric pressure in Denver? PIPE-FLO has a feature that allows for the user to estimate such a value, given an altitude above sea-level. As seen in Figure 3.16, access the *Adjust Document Units* dialog box. Subsequently, click the Calculate Atmospheric Pressure button and enter "1609 m" into the *Elevation* field. Click OK. The estimated atmospheric pressure at that altitude will be displayed in the *Atmospheric Pressure* field.

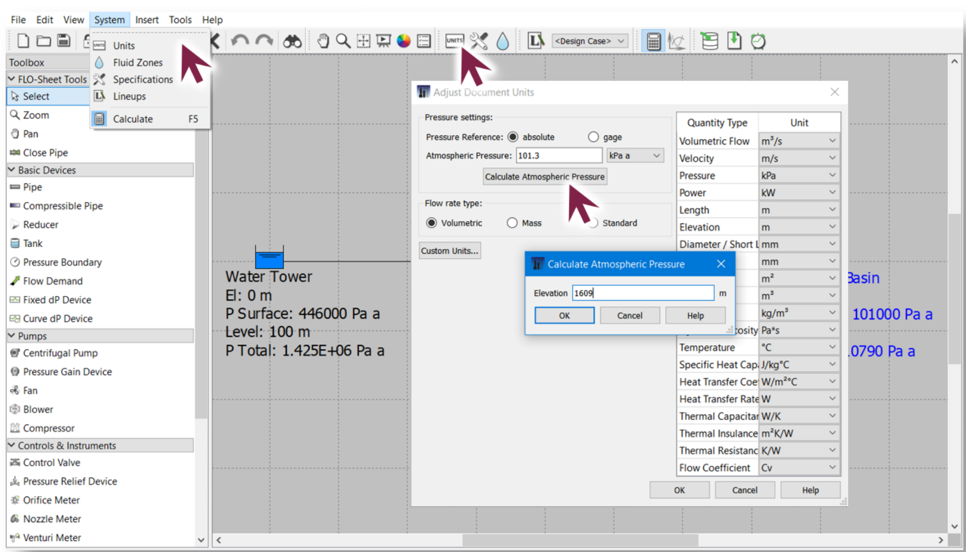


Figure 3.16: *Calculating the estimated atmospheric pressure of a location, given an altitude above sea-level.*

⚠ Do not click OK again. This will save the new atmospheric pressure. Write down the estimated value, then click Cancel.

(B) Return to the FLO-SHEET and set the *Surface Pressure* of both the water tower and receiving basin as the new atmospheric pressure calculated in **Part (A)**, as these tanks are still assumed to be open to the atmosphere. How does the volumetric flow rate, average velocity, and Reynolds Number in the water line now compare to the "base case"?

(C) Verify using Equation 3.2 that the pressure at the bottom of the water tower is 1,062 kPa.

(D) Verify using Equation 3.3 that the Reynold's Number in the water line is 1.817×10^6.

End of Tutorial

Tutorial #4:
Simulation Calculations

Tutorial #4 will consider the following concepts:

- Viewing comprehensive simulation results all at once
- Interpreting simulation results and device properties
- The model equations behind simulation results
- Using the Reynolds Number to identify laminar and turbulent flow
- Head losses associated with piping systems

In the previous tutorial, we highlighted two different methods of viewing device properties and simulation results: directly on the FLO-SHEET and in the *Property Grid*. Let us consider one last method of viewing simulation results— a method particularly useful for viewing a large number of results side-by-side.

List View. This final way of viewing simulation results is through the use of the *List View* window. If it is not already visible on your interface, it can be accessed by navigating to View › List View, or by pressing Ctrl + 3. As seen in Figure 4.1, the window organizes the different types of devices into tabs; in this example, click on one tab to view all the *Pipe* devices or another tab for all the *Tank* devices. All relevant input parameters and simulation results are listed, here. If you find this view useful, the *List View* window can be docked to your interface, as explained in **§1.2 Navigating the PIPE-FLO Interface**.

We will now continue by learning about the different fluid mechanics metrics that PIPE-FLO reports, what they mean, and how they are calculated.

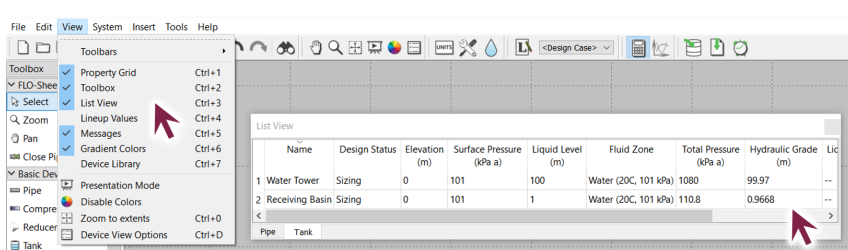


Figure 4.1: *The List View window compiles the simulation results from every device on the* FLO-SHEET *into one handy location.*

Begin by loading the "base case" PIPE-FLO simulation from the previous tutorial—the system described in Figure 3.1. Toggle the *Calculation Mode* to "on" so that the results can be viewed. Your results should match the ones shown in Figure 4.2. If you have not completed Tutorial #3 already, now is a good time to do so.

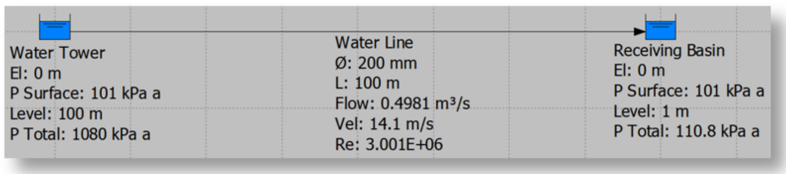


Figure 4.2: *Simulation results for Tutorial #3's water distribution system.*

4.1 Interpreting Simulation Results

In the previous tutorials, we discussed the three ways to display simulation results, but what do these results mean? How do we interpret them? Use this section for reference as we will use these equations later to validate our results. Let us consider the *base case results* from the water tower—a tank, as shown in the *Property Grid* in Figure 4.3.

- *Total Pressure* (FLO-SHEET abbreviation, *P Total*): As mentioned in the previous tutorial, this is the simple static pressure of fluid at the bottom of the tank (P). PIPE-FLO performs this calculation via the basic equation for fluid statics, where P_0 is the *absolute* surface pressure of the fluid, ρ is the fluid density, $g = 9.81$ m/s^2 and h is the height (level) of fluid in the tank.

$$P = P_0 + \rho g h \tag{4.1}$$

- *Hydraulic Grade* (FLO-SHEET abbreviation, *HG*): The hydraulic grade (at the bottom of the tank), HG, is a measure of the potential energy of the fluid in the tank.
- *Liquid Volume* (FLO-SHEET abbreviation, *Vol*): If the *Geometry* of the tank has been specified, this will show the volume of liquid that is in the tank at the beginning of the simulation.

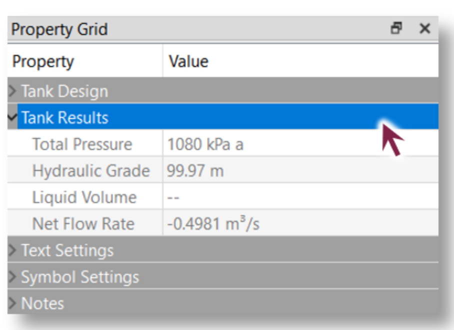


Figure 4.3: *Simulation results from the water tower in Tutorial #3.*

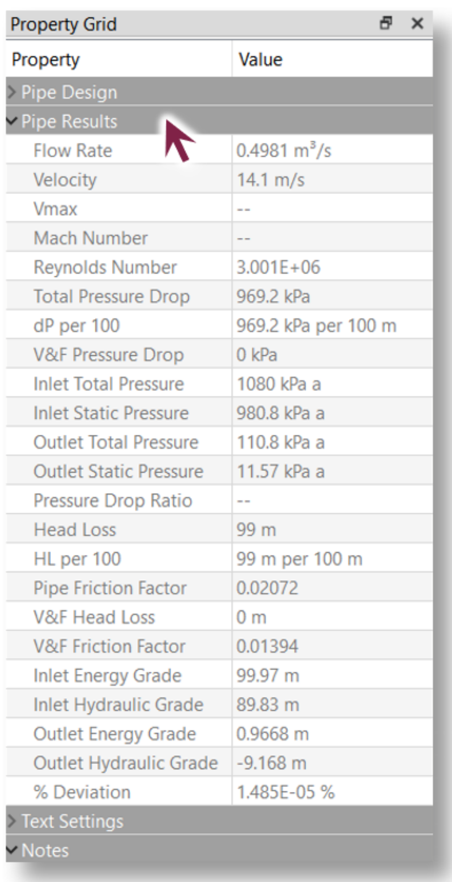
- *Net Flow Rate* (Flo-Sheet abbreviation, *Net Flow*): This displays the net flow rate entering or exiting a tank. The negative sign indicates that fluid is exiting the tank.

The simulation results for the water line—a pipe—contains a wealth of information about the system at steady-state. 21 results are shown in the *Property Grid*, as seen in Figure 4.4.

- *Flow Rate* (abbr. *Flow*): The calculated flow rate Q through the pipe. This value is obtained by performing an iterative "network analysis" on the piping system model until the conservation of mass and conservation of energy balances in the system are satisfied.
- *Velocity* (abbr. *Vel*): The *average* velocity of fluid ($v = v_{\text{avg}}$) flowing through the pipe. This is calculated using the relationship between the pre-calculated volumetric flow rate Q and the pipe inner diameter D, assuming a circular pipe cross-sectional area, A:

$$v = \frac{Q}{A} = \frac{4Q}{\pi D^2} \tag{4.2}$$

- *Vmax* (abbr. *Vmax*): The velocity at the maximum in the stock friction loss curve for pulp fluids. We will not use this parameter.
- *Mach Number* (abbr. *Ma*): The Mach Number is a parameter associated with compressible fluids, calculated as a ratio of the fluid velocity to the speed of sound in the fluid.


Property Grid

Property	Value
Pipe Design	
Pipe Results	
Flow Rate	0.4981 m³/s
Velocity	14.1 m/s
Vmax	--
Mach Number	--
Reynolds Number	3.001E+06
Total Pressure Drop	969.2 kPa
dP per 100	969.2 kPa per 100 m
V&F Pressure Drop	0 kPa
Inlet Total Pressure	1080 kPa a
Inlet Static Pressure	980.8 kPa a
Outlet Total Pressure	110.8 kPa a
Outlet Static Pressure	11.57 kPa a
Pressure Drop Ratio	--
Head Loss	99 m
HL per 100	99 m per 100 m
Pipe Friction Factor	0.02072
V&F Head Loss	0 m
V&F Friction Factor	0.01394
Inlet Energy Grade	99.97 m
Inlet Hydraulic Grade	89.83 m
Outlet Energy Grade	0.9668 m
Outlet Hydraulic Grade	-9.168 m
% Deviation	1.485E-05 %
Text Settings	
Notes	

Figure 4.4: *Simulation results for the water line from Tutorial #3.*

- *Reynolds Number* (abbr. *Re*): The Reynolds Number Re is used characterize the flow in a pipe as laminar or turbulent. It is a function of the pipe's inner diameter D, and the fluid velocity v, density ρ, and absolute viscosity μ.

$$Re = \frac{\rho v D}{\mu} \tag{4.3}$$

- *Total Pressure Drop* (abbr. *dP Total*): The pressure drop ΔP is the sum of pressure losses across the pipe of constant diameter due to major frictional losses, minor losses from valves and fittings, as well as elevation changes over the total length of the pipe. ΔP can also be ascertained from a rearrangement of the Bernoulli Equation (to be seen in Eqn. 4.12), given the total head loss H_{l_T}, and the pipe inlet and outlet elevations, z_{in} and z_{out}. This assumes no change in volumetric flow rate.

$$\Delta P = \rho g H_{l_T} + \rho g (z_{\text{out}} - z_{\text{in}}) \tag{4.4}$$

- *dP per 100* (abbr. *dP/100*): Whereas the *pressure drop* value refers to the pressure lost over the entire pipe, this value refers to pressure lost per 100 units of length (m, in our case).
- *V&F Pressure Drop* (abbr. *V&F dP*): The "valves and fittings" pressure drop is due to the minor losses inherent with valves, fittings, tank entrances and exits. This is calculated from the minor losses portion of the Bernoulli Equation. The calculation for minor losses is shown in Equation 4.11.
- *Inlet/Outlet Total Pressure* (abbr. *P in Total/P out Total*): The inlet/outlet pressures, P_{in} or P_{out}, refer to the *actual hydrostatic pressure* of the fluid at the inlet or outlet of the pipe (*i.e* the pressure of the fluid in the tank at the pipe inlet/outlet.
- *Inlet/Outlet Static Pressure* (abbr. *P in Static/P out Static*): The *inlet/outlet static pressures*, $P_{\text{static, in}}$ or $P_{\text{static, out}}$ are components of the *inlet/outlet pressures*, P_{in} or P_{out}, and must not be confused.
- *Pressure Drop Ratio* (abbr. *PDR*): This parameter is associated with compressible fluids. We will not use this parameter.
- *Head Loss* (abbr. *HL*): The total head loss H_{l_T} is the sum of head losses across the pipe major frictional losses H_l and minor losses H_{l_m} due to bends, valves, fittings, entrances and exits. Expressed in units of length, the head loss is:

$$H_{l_T} = H_l + H_{l_m} \tag{4.5}$$

- *HL per 100* (abbr. *HL/100*): Whereas the *head loss* value refers to the head lost over the entire pipe, this value refers to head lost per 100 units of length (m, in our case).
- *Pipe Friction Factor* (abbr. *ffp*): In regimes of *turbulent flow*, the Darcy-Weisbach friction factor f is calculated in Pipe-Flo via the Serghide Equations (below) in order to obtain a value for the major head loss in the pipe, given the pipe absolute roughness e and inner diameter D

$$f = \left(A - \frac{(B - A)^2}{C - 2B + A} \right)^{-2} \tag{4.6a}$$

Where:

$$A = -2\log\left(\frac{e/D}{3.7} + \frac{12}{Re}\right) \tag{4.6b}$$

$$B = -2\log\left(\frac{e/D}{3.7} + \frac{2.51A}{Re}\right) \tag{4.6c}$$

$$C = -2\log\left(\frac{e/D}{3.7} + \frac{2.51B}{Re}\right) \tag{4.6d}$$

The *relative roughness* is often used to relate e and D, replacing these values in the equations for f. The relative roughness ε is defined as:

$$\varepsilon = \frac{e}{D} \tag{4.7}$$

- *V&F Head Loss* (abbr. *V&F HL*): The "valves and fittings" head loss is attributed to the minor losses inherent with valves, fittings, tank entrances and exits.
- *V&F Friction Factor* (abbr. *fft*): The "valves and fittings" friction factor is used in very specific cases to calculate the head loss associated with the valves and fittings installed in the pipe. We will not observe this value.
- *Inlet/Outlet Energy Grade* (abbr. *EG in/EG out*): The inlet/outlet energy grades (EG_{in} or EG_{out}) are the total hydraulic potential energies of the fluid at the inlet or outlet of the pipe, using the atmospheric pressure (P_{atm}) and the elevation of the inlet/outlet (z_{in} or z_{out}) as reference points. We will not consider this parameter in this course.
- *Inlet/Outlet Hydraulic Grade* (abbr. *HG in/HG out*): The inlet/outlet hydraulic grades (HG_{in} or HG_{out}) are the total *static* heads (*i.e.* from the static pressures) of the fluid at the inlet/outlet of the pipe, using the atmospheric pressure (P_{atm}) and the elevation of the inlet/outlet (z_{in} or z_{out}) as reference points.
- *% Deviation* (abbr. *% Deviation*): PIPE-FLO states that the percent deviation is the difference between a device's pressure drop calculated by PIPE-FLO's iterative method, and the device's pressure drop calculated with the device's performance equation/graph at PIPE-FLO's calculated flow rate. A small percent deviation is indicative of good model convergence.

4.2 Laminar and Turbulent Flow

In Tutorial #3, we used the formula for the Reynold's Number in a circular pipe (*see* Equations 3.3 and 4.3) to verify PIPE-FLO's calculations in the water distribution system. The Reynolds Number is particularly useful to us as it can be used to describe the *regime* in which the flow within the pipe resides: *laminar* or *turbulent*. Subsequently, knowing the flow regime is crucial for determining the *friction factor* associated with a pipe. Let us proceed to investigate laminar and turbulent flow.

Return to the "base case" FLO-SHEET from Tutorial #3. Now is a good time to make a copy of that file, and save it under a new name—Tutorial #4. Check back to Figure 4.2 to ensure that you have the same results.

Laminar Flow. Let us proceed with a few modifications to our water distribution system. Suppose that the year is particularly dry, and the water level in the tower has fallen drastically, yet the level in the receiving basin remains constant. Furthermore, a piping retrofit project has changed the material, diameter, and length of the water line. Make the following changes to the FLO-SHEET:

- Change the water level in the water tower to 5 m.
- Create a new specification for Schedule 80 PVC plastic pipe.
- Change the water line to be made of this PVC plastic pipe, with a much lower nominal diameter of 10 mm.
- Finally, increase the length of the water line to 1,000 m. Leave everything else as-is.

The simulation results should appear identical to those seen in Figure 4.5. Observe that the Reynolds Number in this simulation is $Re = 1532$. We can confirm this value ourselves using Equation 4.3 and the following information:

- The fluid density and viscosity (from the *Fluid Zones* dialog box) are $\rho = 998.3$ kg/m^3 and $\mu = 9.946\times10^{-4}$ Pa·s.
- The fluid velocity is $v = 0.142$ m/s.
- The pipe inner diameter (from the *Property Grid*) is $D = 10.74$ mm.

$$Re = \frac{(998.3\ \mathrm{kg/m^3})(0.142\ \mathrm{m/s})(10.74\times10^{-3}\ \mathrm{m})}{(9.946\times10^{-4}\ \mathrm{Pa\cdot s})}$$

$$\boxed{Re = 1531}$$

Our manually-calculated Reynolds Number agrees well with value determined by PIPE-FLO. Determining the Reynolds Number helps to evaluate the flow regime:

$$\begin{cases} Re < 2300: & \text{Laminar Flow} \\ 2300 < Re < 4000: & \text{Mixed Flow} \\ Re > 4000: & \text{Turbulent Flow} \end{cases} \tag{4.8}$$

Therefore, the flow in the water line is clearly in the *laminar regime*.

Here is one last reminder to set your *Laminar Cutoff Re* to 2300—it starts to matter now that we are operating in a laminar regime. Remember that this value can be set in the *Property Grid* under *Calculation Settings*.

In the laminar flow regime within a circular conduit, PIPE-FLO does not actually calculate the laminar Darcy-Weisbach friction factor f_{lam} using the Haaland Equation (Eqn. 2.2), nor even the Serghide Equations (Eqn. 4.6a); PIPE-FLO uses the simple expression:

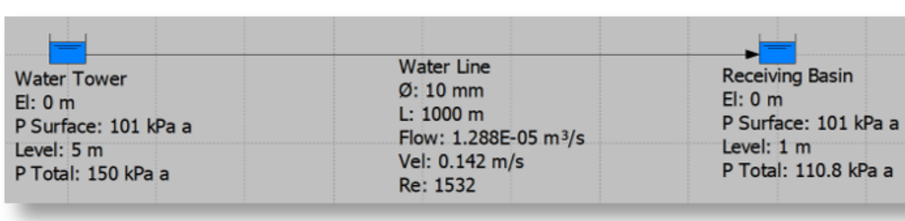


Figure 4.5: *Simulation results from the modified water distribution system, resulting in laminar flow.*

$$f_{\text{lam}} = \frac{64}{Re} \tag{4.9}$$

We can verify this by using the simulation results in Figure 4.5, with $Re = 1532$. PIPE-FLO calculated that the pipe friction factor is $f_{\text{lam}} = 0.04178$.

$$f_{\text{lam}} = \frac{64}{1532}$$

$$\boxed{f_{\text{lam}} = 0.04178}$$

Turbulent Flow. Now suppose that the year's dry streak is ending and the water level in the tower rises to 30 m. Change the level to this value and observe the effect on the Reynolds Number. As expected, the Re rises due to the increase in hydrostatic pressure in the tower, and causing the fluid velocity in the pipe to rise as well. Observe the results in Figure 4.6. Now, the Reynolds Number is $Re = 4250$, so the flow enters the *turbulent regime*. As mentioned previously, the turbulent friction factor f is calculated using the Serghide Equations (Eqn. 4.6a). We will not perform this verification in its full length as the sequential calculation of A, B, C, and f becomes tedious.

Calculating the friction factor by-hand using the Serghide Equations is certainly doable, but is made much easier through a spreadsheet. Should you try to verify the calculation of f yourself, these are the values that you should get:

$$A = 5.08670$$

$$B = 5.03354$$

$$C = 5.04255$$

$$\boxed{f = 0.03935}$$

We found that $f = 0.03935$. Notice that this is a higher friction factor than f_{lam}—an intuitive result.

Mixed Flow. Suppose that the water level in the tower decreases back down to 15 m. In such a circumstance, we encounter an example of *mixed flow*. Observe in Figure 4.7 that the Reynolds Number is $Re = 2770$ and accordingly, $2300 < Re < 4000$, indicating mixed flow.

In cases of mixed flow, PIPE-FLO treats the flow regime as "turbulent"; thus, the Serghide Equations are used.

Applying the Serghide Equations again, we find that $f = 0.04473$, with $A = 4.71894$, $B = 4.73017$, and $C = 4.72812$.

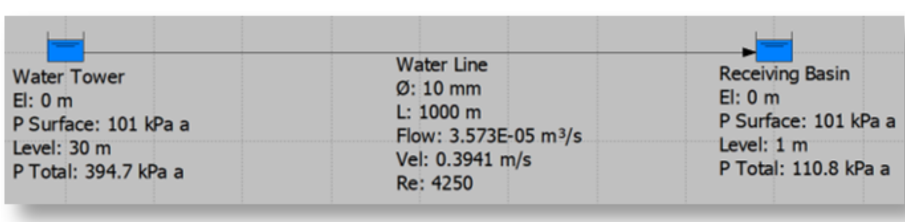


Figure 4.6: *Simulation results for the modified water distribution system with a level of 30 m in the Water Tower, resulting in turbulent flow.*

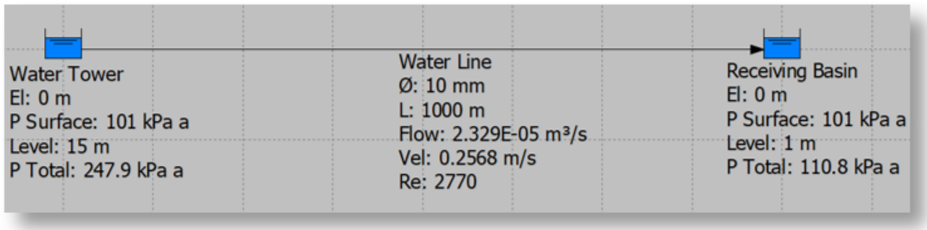


Figure 4.7: *Simulation results for the modified water distribution system with a level of 15 m in the Water Tower, resulting in mixed flow.*

For most cases in piping systems, the flow regime is turbulent, therefore the Serghide Equations are valid. Only in limited scenarios, such as with extremely viscous fluids (*e.g.* tars, molasses, polymer melts, etc.), very low densities (*e.g.* low-pressure gases) or low flow velocities do systems leave the turbulent regime. Always keep the flow regime in mind when using Pipe-Flo!

4.3 Head Losses

The *head* of a particular system at a specified location is a measure of the amount of energy present. For example, if it is stated that there is *100 m of head* available at the bottom of a water tank, physically, this means that there is enough energy (*i.e.* pressure) to transport the water directly upwards against gravity to a point exactly *100 m higher* than the bottom of the tank. This is provided that no losses occur. Whereas the *system head* is the amount of energy *present*, the *head loss* is the total *reduction in system head* caused by friction against the pipe, as well as from other components in the system. Heads can be expressed in two different forms with two different sets of units:

- A head expressed as an uppercase 'H'—such as H_{l_T}, H_l, H_{l_m}—refers to a value expressed in the aptly-named *head form*, with units of *length* (m).
- A head expressed as a lowercase 'h'—such as h_{l_T}, h_l, h_{l_m}—refers to losses expressed in *standard form*, with units of *length²/time²* (m²/s²).

To convert between the two forms, the following simple expression can be used, where g is the gravitational constant ($g = 9.81$ m/s²):

$$H = \frac{h}{g} \tag{4.10}$$

The total head loss H_{l_T} is comprised of major (frictional) losses H_l as well as minor losses H_{l_m} due to bends, valves, fittings, and other components:

$$H_{l_T} = \underbrace{f \frac{L}{D_h} \frac{v_{\text{avg}}^2}{2g}}_{H_l} + \underbrace{\sum \left(K \frac{v_{\text{avg}}^2}{2g} \right)}_{H_{l_m}} \tag{4.11}$$

Observe that the frictional losses are dependent on pipe characteristics (friction factor f, length L, hydraulic diameter D_h) and the fluid velocity v_{avg}, while the minor losses are only a function of the fluid velocity and the sum of the individual loss (resistance) coefficients $\sum K$. Pipe-Flo uses the Bernoulli Equation with the term for total head losses in order to calculate its results:

$$\frac{P_1}{\rho g} + \alpha_1 \frac{v_1^2}{2g} + z_1 = \frac{P_2}{\rho g} + \alpha_2 \frac{v_2^2}{2g} + z_2 + H_{l_T} \tag{4.12}$$

Notice that Equations 4.11 and 4.12 are written in the *head form.* While the standard form of these equations may also be used, it is inherently more useful to write the equations in the head form (with units of length) as PIPE-FLO reports major, minor and total losses as heads. While PIPE-FLO uses this convention, you should make yourself comfortable with converting between the two forms.

Let us consider the major (frictional) losses in our modified water distribution system. Recall the results in Figure 4.7: flow in the water line was in the mixed flow regime. The following data was also recorded:

- The pipe friction factor f is 0.04473.
- The pipe length L is 1,000 m.
- The average flow velocity v_{avg} is 0.2568 m/s.
- The hydraulic diameter (*i.e.* inner wetted diameter) D_h of the pipe is 10.74 mm.
- The total head loss H_{l_T} is 14 m.
- No minor losses were recorded. The valves and fittings head loss H_{l_m} is 0 m.

Using Equation 4.11, we find:

$$H_{l_T} = \underbrace{(0.04473)\frac{1000 \text{ m}}{10.74 \times 10^{-3} \text{ m}} \times \frac{(0.2568 \text{ m/s})^2}{2 \times (9.81 \text{ m/s}^2)}}_{H_l} + \underbrace{\frac{0}{2 \times (9.81 \text{ m/s}^2)}}_{H_{l_m}}$$

$$\boxed{H_{l_T} = 14.0 \text{ m}}$$

Our value calculated by-hand is identical to the value calculated by PIPE-FLO, seen in Figure 4.8!

One final note with regards to calculating major losses: PIPE-FLO can actually employ *two* different calculation methods in order to arrive at the final result. **(A)** The *Darcy-Weisbach* method uses the Darcy-Weisbach friction factor (f) in order to calculate major losses. This is the method that we use, and it applies to almost all fluids. **(B)** The *Hazen-Williams* method uses the Hazen-Williams frictional coefficient (C) in order to calculate major losses. This method only works well for water, and it is beyond the scope of these tutorials. Ensure that you are using the Darcy-Weisbach method by clicking anywhere on the FLO-SHEET, then navigating to the *Calculation Settings* heading in the *Property Grid*. Ensure that the *Calculation Method* reads "Darcy-Weisbach"—also shown previously in Figure 1.7.

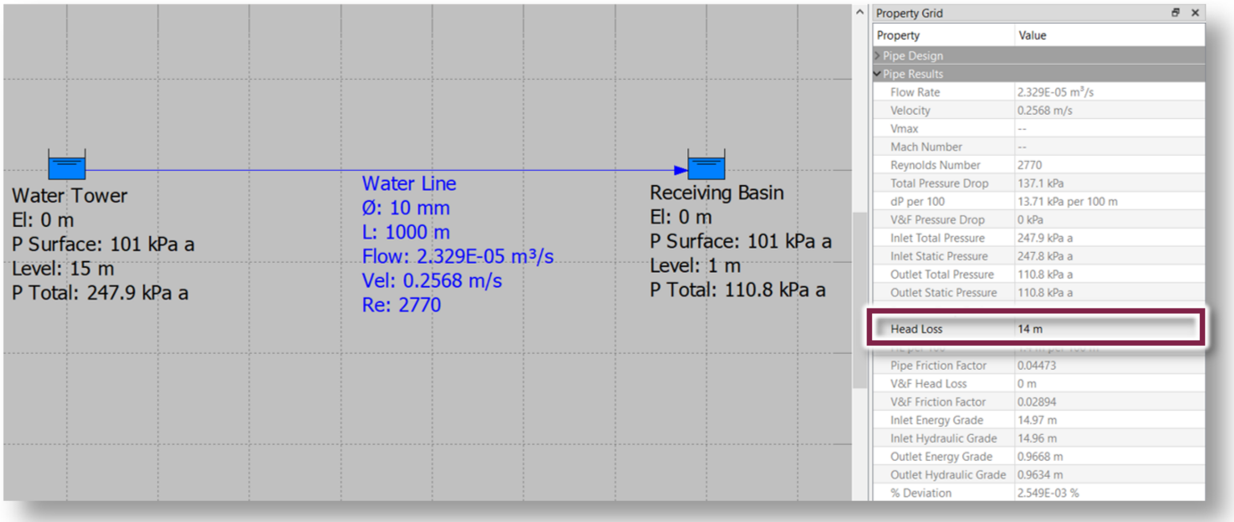
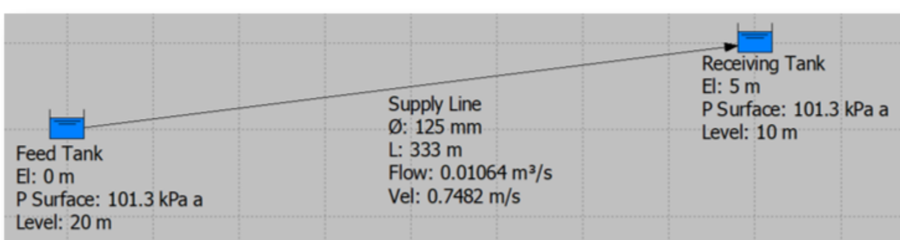


Figure 4.8: *The total head loss in the modified water distribution system.*

Tutorial Problems

Problem #1. Consider the simple fluid statics system modeled in PIPE-FLO, shown below. Answer the questions regarding the system. Everything you need is shown on the FLO-SHEET, in addition to the following data:

- The fluid density is 1134 $\mathrm{kg/m^3}$ and the fluid viscosity is 0.1263 $\mathrm{Pa \cdot s}$.



(A) Given the volumetric flow rate and average fluid velocity, calculate the inner diameter of the supply line pipe.

(B) Calculate the Reynolds Number of the fluid passing through the supply line.

(C) What type of flow is occurring in the pipe?

(D) Calculate the pipe's friction factor using the appropriate equation.

(E) Calculate the major (frictional) losses in the pipe as well as the total head loss. Note that there are no components that cause a minor loss.

(F) Given the total head loss, calculate the pressure drop across the supply line.

(G) What are the hydrostatic pressurse at the bottom of the two tanks?

(H) How does the *difference* in tank pressures ($P_{\text{feed}} - P_{\text{receiving}}$) from Part **(G)** compare to your answer from Part **(F)**? Use Equation 4.4 or 4.12 as a justification.

Problem #2. Build the Flo-Sheet from Problem #1 to compare the simulated results to the values produced by the manual calculations.

(A) Create a new Flo-Sheet. Set a fluid zone for ethylene glycol at -10°C and 101.325 kPa. Also, create a piping specification for standard (STD) schedule B302 copper pipe.

(B) Add the two tanks and the pipe connecting them according to the Flo-Sheet from Problem #1. As usual, the penetrations are at the bottoms of the tanks.

(C) Run your simulation. Record the values of the:

- Inner pipe diameter;
- Reynolds Number;
- Pipe friction factor;
- Major and total head losses;
- Pressure drop across the pipeline; and
- Hydrostatic pressure at the bottom of the two tanks.

How do these values compare to Problem #1?

Save your Flo-Sheet as it will be used in the next problem!

Problem #3. Using the Flo-Sheet created in the previous problem, make the following changes to the *process fluid* and *piping specifications*. Record the effect of these changes on the indicated process parameters.

(A) What is the effect of the refrigerant properties (at -10°C and 101.325 kPa) in the table (below) on the system? Use the fluid densities and viscosities to make comparisons.

(B) Choose any refrigerant from the table. Record the corresponding volumetric flow rate Q along with the copper pipe's absolute roughness e.

(C) Now, create a pipe specification for standard (STD) schedule iron pipe and apply that specification to the supply line. Leave the nominal diameter ∅ at 125 mm.

(D) Record the new volumetric flow rate Q along with the iron pipe's absolute roughness e. What do you notice?

Refrigerant	Fluid Density (kg/m^3)	Fluid Viscosity (Pa·s)	Reynolds Number	Flow Regime	Pipeline Pressure Drop (Pa)	Head Loss (m)
Ethylene Glycol						
R113						
R-600						

Pipe Specification	Fluid	Volumetric Flow Rate (m^3/s)	Absolute Roughness (mm)
Copper Pipe B302			
Iron Pipe			

End of Tutorial

Tutorial #5:
Fluid Statics with Minor Losses

Tutorial #5 will consider the following concepts:

- Adding valves, fittings and other components that create minor losses
- Compensating for tank entrance and exit losses
- Specifying the degree to which a valve is open or closed
- Splitting pipes and creating nodes

The frictional losses associated with transporting fluid through a "rough" pipe are considered as "major" losses in the system as they can often represent a large fraction of the total losses. However, do not let this fool you into thinking that "minor" losses due to valves, fittings, entrances, exits, and other components are insignificant. In some piping systems, these components can sometimes produce a head loss even greater than the major head loss!

5.1 Simulating Minor Losses

In this section of the tutorial, we will simulate the simple pressure-driven movement of liquefied propane gas (LPG) at 15°C between pressurized cylinders (*i.e.* a feed tank and a receiving tank), as seen in Figure 5.1. Both tanks have a surface pressure of 1 MPa while the LPG levels in the feed and and receiving tanks are 35 and 15 m, respectively. Furthermore, the feed tank sits 75 m higher than the receiving tank. These cylinders are separated by a 100 m long (in total), 80 mm nominal diameter Schedule 80S stainless steel ASME B36.19M pipeline with two standard 90° elbows. There are further losses associated with the "inward pipe entrance" and "projecting pipe exit" connections with the cylinders. As PIPE-FLO can only simulate *steady-state systems*, we will again assume that the *cylinders are being fed at the same rate that they are being drained*. We will investigate this scenario in the following tutorial.

Let us modify our water delivery system from the previous tutorial to model this scenario. We will make some substantial changes to the system, so it is recommended that you duplicate the "base case" scenario and use the second copy, here. Alternatively, you could start from scratch for the practice.

With your unmodified "base case" file open, navigate to File › Save as..., or alternatively, type Ctrl + Shift + S to save a duplicate of the file.

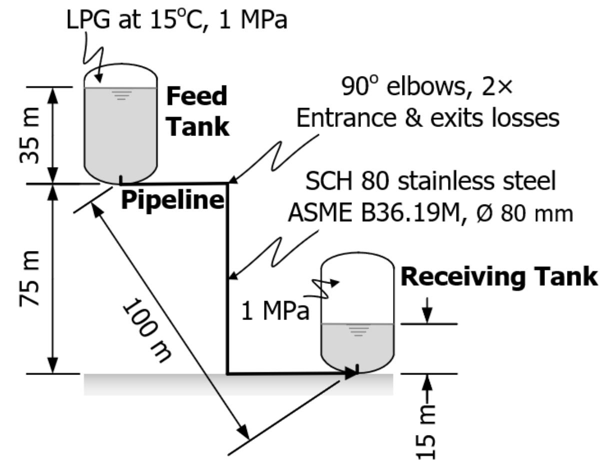


Figure 5.1: *LPG feed and receiving tanks connected by a pipe.*

When making significant changes to the FLO-SHEET such as adding/removing devices, it is almost guaranteed that PIPE-FLO will experience errors while *Calculation Mode* is enabled due to equipment that is not fully-specified. These errors can be distracting for the user. It is therefore recommended that *Calculation Mode* be turned off for major changes, such as right now. You will be informed when it is time to restart *Calculation Mode*.

With the duplicate file open, we will start by making the following changes, illustrated in Figure 5.2:

- Delete the water line. This can be done by clicking on the pipe and then typing Delete or by clicking on the pipe and then clicking on the *Delete* icon (✕).
- Create a fluid zone for LPG at 15°C and 1 MPa.
- Create a specification for Schedule 80S stainless steel ASME B36.19M pipe.
- Move your first tank (the one labeled "Water Tower") to a higher location on the FLO-SHEET as it is at a higher elevation in the scenario described earlier. This can be done by clicking and dragging it, or by clicking it and using the arrow keys on your keyboard.

Remember that merely moving the tank to a higher position does not tell PIPE-FLO anything about the elevation of the tank. We will have to set the elevation later.

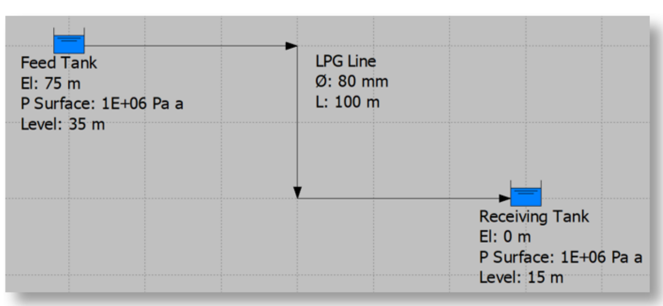


Figure 5.2: FLO-SHEET *of the LPG feed and receiving tanks connected by a pipe.*

- Join the two tanks together with a pipe. To get the "bends", as seen in Figure 5.2, start by selecting the *Pipe* tool. Click on the upper tank, and drag your cursor to where you would like the first bend to appear. Click again, and drag your cursor to the next location, and so on. The pipe will continue to be drawn until (1) the user *double-clicks*, specifying the end of the pipe (a *node*), or (2) the user clicks on a tank or other device to which a pipe may be connected.

> Creating "bends" in a pipe does not indicate to PIPE-FLO that there is a valve or fitting at that location. The user must subsequently enter information regarding valves and fittings, as we will later.

- For the upper tank, set its *Name* to "Feed Tank", its *Elevation* to 75 m, its *Surface Pressure* to 1 MPa, its *Liquid Level* to 35 m, and its *Fluid Zone* to LPG, as specified in the description.
- For the lower tank, set its *Name* to "Receiving Tank", its *Elevation* to 0 m, its *Surface Pressure* to 1 MPa, its *Liquid Level* to 15 m, and its *Fluid Zone* to LPG, as specified in the description.
- For the pipe, set its *Name* to "LPG Line", its *Fluid Zone* to LPG, its *Specification* to the Schedule 80S stainless steel ASME B36.19M pipe, its *Size* to 80 mm, and its *Length* to 100 m. Everything else can be left as-is.

We are now ready to simulate, again. Turn on *Calculation Mode*. Your FLO-SHEET should appear similar to the one shown in Figure 5.3. To ensure that you are on the right track, you may choose to set your *Device View Options* to match the properties shown in this figure. Note the following pipe properties, in the absence of minor losses:

- The volumetric flow rate Q is 0.03754 m^3/s;
- The average velocity of fluid v_{avg} is 8.809 m/s; and
- The Reynolds Number Re is 3.074×10^6.

> You may have noticed that once you placed the bends in your pipe, the text that accompanies it becomes rotated by 90°. To remedy this situation, click on the pipe, then find the *Text Settings* heading in the *Property Grid* (under *Pipe Results*). Under this heading, replace "Smart" with "Horizontal" in the *Text Orientation* drop-down menu. This can be seen in Figure 5.4.

Adding Components with Minor Losses. Now, we will add our minor losses (elbows, entrance/exit losses) to the FLO-SHEET. To do this, take the following steps, illustrated in Figure 5.5. *Calculation Mode* may remain on.

1. Select the pipe (LPG Line).

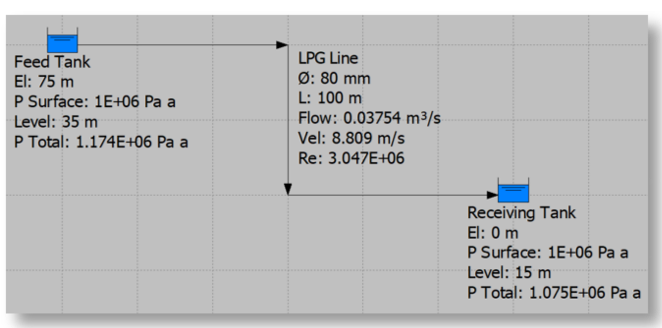


Figure 5.3: *Simulation for the LPG feed and receiving tanks connected by a pipe with no minor losses.*

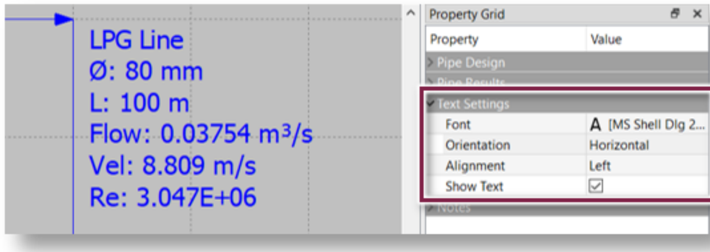


Figure 5.4: *Reverting the LPG Line's descriptive text to the horizontal orientation.*

2. In the *Property Grid* under the *Pipe Design* heading, click the "--" placeholder in the *K (Valves & Fittings)* field. *K* this is the value of the *resistance coefficient*. This opens the *Valves and Fittings* dialog box. Here, we can select the components that will generate minor losses. In our case, we have two standard 90° elbows, one inward entrance, and one projecting exit.

3. Start with the elbow joints. Under the "*Fitting*" heading in the menu on the right side of the dialog box, click on "Elbow - Standard 90°" from the list. Under the *Settings* panel, type "2" into the *Count* field, specifying that there are two elbows in the pipe. (Notice the *K* value for these two fittings is displayed!) You may also choose name the fitting in the *Name* field. Click the Add button to add these fittings to the pipe.

4. Continue with the entrance loss. Under the "*Fitting*" heading, select "Entrance - Inward" from the list. Ensure that the *Count* field reads "1", then click the Add button.

5. Finally, we will add the exit loss. Still under the "*Fitting*" heading, select "Exit - Projecting" from the list. Ensure that the *Count* field reads "1", then click the Add button.

6. If you make a mistake when adding valves and fittings, select the offending component from the "*Installed*" panel, then press the Delete button.

When you are finished, press OK. The (total) minor loss coefficient $\sum K$ will be displayed in the *K* (*Valves & Fittings*) field. As well, there will be a small elbow ( ) symbol located on the LPG line, denoting that there are

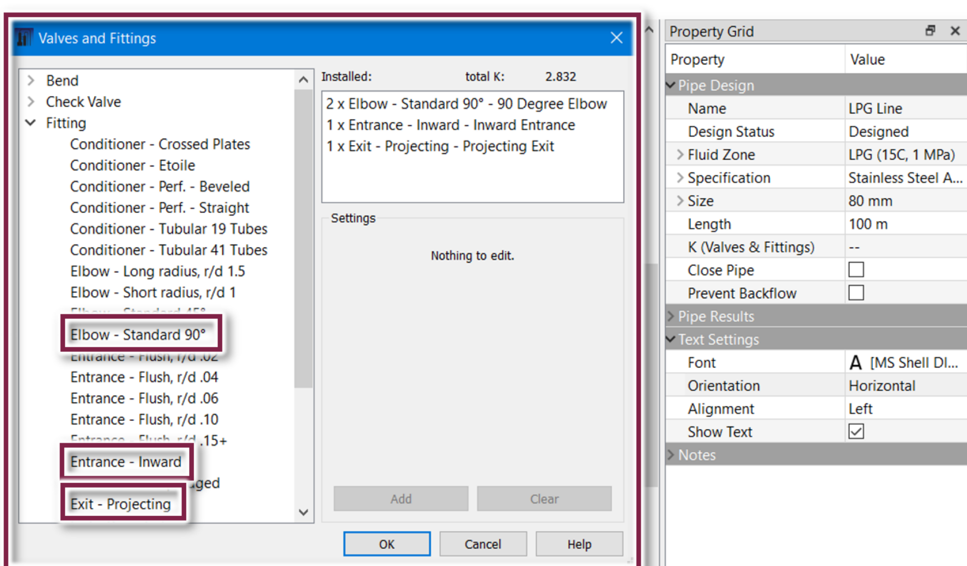


Figure 5.5: *Selecting components with minors losses to be added to the* Flo-Sheet.

minor losses due to fittings on that pipe. This is seen in Figure 5.6. Check the results of the ongoing simulation: how do they compare to the system with no minor losses?

- The volumetric flow rate Q was 0.03754 m^3/s. It is now 0.0355 m^3/s
- The average velocity of fluid v_{avg} was 8.809 m/s. It is now 8.329 m/s.
- The Reynolds Number Re was 3.074×10^6. It is now 2.881×10^6.
- The total minor loss coefficient ($\sum K$) was non-existent (or zero), previously. It is now 2.832.

The inclusion of components with minor losses such as elbows and tank entrances/exits adjusts the results of the simulation to be more accurate. It is clear that adding these components increases the head loss across the pipe, decreasing the volumetric flow rate and fluid velocity, and thereby reducing the Reynolds Number.

Valves and Minor Losses. A valve—whether open or partially-closed—creates resistance to fluid flow. We can use this principle to actually control the flow in our LPG pipe.

Suppose operators at the receiving tank request a flow rate of 0.01 m^3/s of LPG. We can use a globe valve—seen in Figure 5.7—on the LPG pipe to restrict the flow rate through this valve:

1. Re-enter the *Valves and Fittings* dialog box by clicking on the $\sum K$ value in the *K (Valves & Fittings)* field.
2. Under the *Valve* heading, we will select a standard globe valve, as seen in Figure 5.8. Click on "Globe".
3. In the *Settings* panel, PIPE-FLO asks for a *reduced seat diameter* (abbreviated to "RSeat dia"): this is a metric which describes how far open/closed the valve is. The reduced seat diameter *must not be larger than the inner diameter of the pipe*, itself. A reduced seat diameter of *I.D.* mm represents a fully-open (*fully-seated*) valve, while decreasing this value increases the degree of valve closure. The reduced seat diameter must be *greater than 0 mm*, otherwise the pipe would be fully-closed. For now, we will design our valve to have a reduced seat diameter of 50 mm.
4. Add a *Name* if you wish, ensure the *Count* field reads "1", and click Add. Return to the FLO-SHEET.

> ℹ Leaving the *RSeat dia.* field blank refers to a fully open valve. Notice how the K value increases with decreasing reduced seat diameter?

Notice that a valve icon ()has appeared on the LPG line to indicate that there a is a valve contributing to the minor losses in the pipe. Now compare the results seen in Figure 5.9 with the results from before the valve was installed:

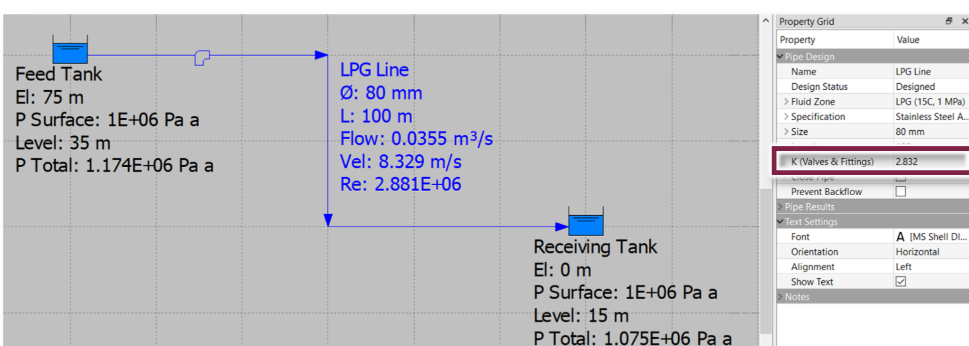


Figure 5.6: *The* FLO-SHEET *now has minor losses on the LPG Line, totaling 2.832.*

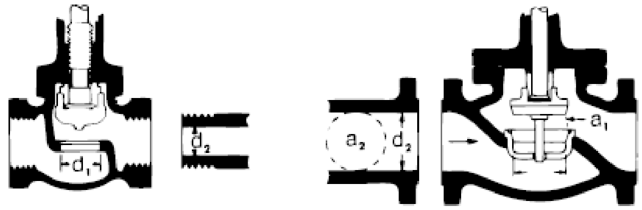


Figure 5.7: *Schematic of two standard globe valves.*

- The volumetric flow rate Q decreased from 0.0355 to 0.02384 m^3/s.
- The average velocity of fluid v_{avg} decreased from 8.329 to 5.595 m/s.
- The Reynolds Number Re decreased from 2.881×10^6 to 1.936×10^6.
- The total minor loss coefficient $(\sum K)$ was 2.832 and has drastically increased to 35.4.

Clearly, adding a partially-closed valve increases the minor loss coefficient, which increases the head loss across the pipe, decreasing the volumetric flow rate and fluid velocity, which reduces the Reynolds Number. However, we have not yet achieved the required flow rate of 0.01 m^3/s. Return to the *Valves and Fittings* dialog box, and *increase* the resistance caused by the globe valve by *decreasing* the reduced seat diameter. Click on your *Globe Valve* in the *Installed* panel in order to edit the reduced seat diameter; click Save to make the change.

Iteratively change the reduced seat diameter until you reach a flow rate of approximately 0.01 m^3/s. What is the reduced seat diameter and total minor loss coefficient $(\sum K)$ that achieves this flow rate?

If you iterate for long enough, you will discover that a reduced seat diameter of 29–30 mm will produce an LPG flow rate of approximately 0.01 m^3/s. This is with a total minor loss coefficient ranging from 286.1–330.1. The results are shown in Figure 5.10.

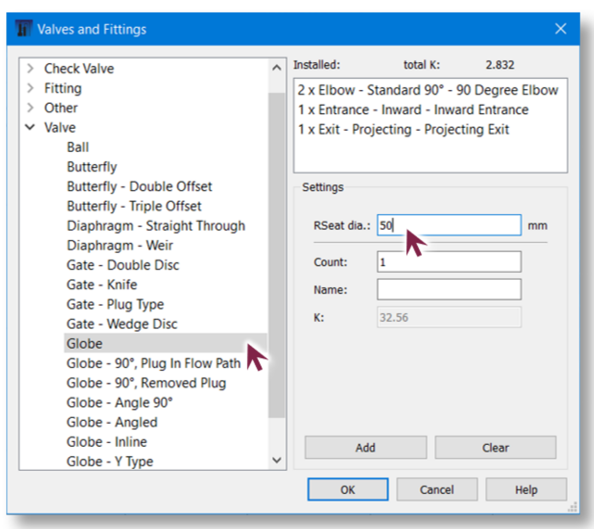


Figure 5.8: *Adding a globe valve with a reduced seat diameter of 50 mm.*

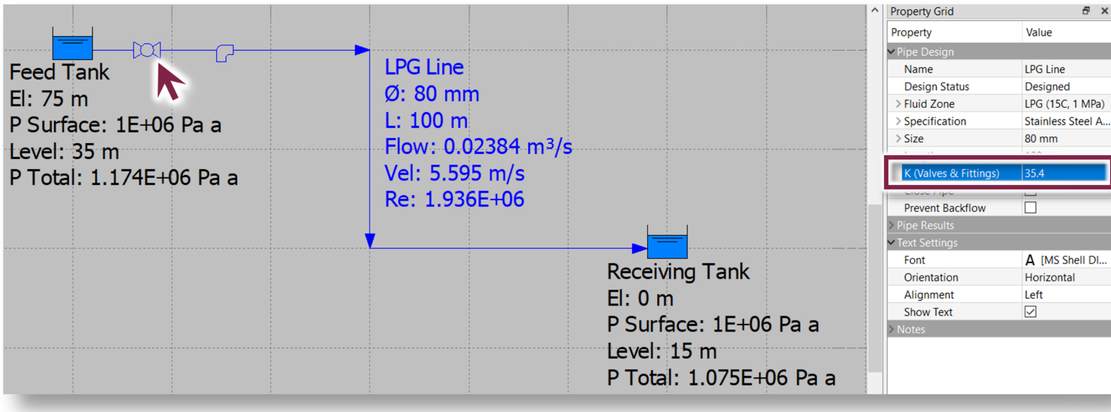


Figure 5.9: *The* FLO-SHEET *with minor losses from entrances/exits, elbows, and a globe valve.*

Set the reduced seat diameter of your LPG Line to be 29.5 mm, and re-run the simulation to match the results from this tutorial.

Manual Verification of Losses. Let us use this opportunity to manually verify PIPE-FLO's calculations with regards to minor losses in this fluid statics system. Recall from Equation 4.11 in the previous tutorial:

$$H_{l_T} = \underbrace{f \frac{L}{D_h} \frac{v_{\text{avg}}^2}{2g}}_{H_l} + \underbrace{\sum \left(K \frac{v_{\text{avg}}^2}{2g} \right)}_{H_{l_m}}$$

Let us calculate the major, minor and total losses in this system thus far. To calculate the major frictional losses, we employ the H_l term from the above equation. We can find the values of the variables from the data that we have input into PIPE-FLO, as well as from the simulation results.

- The pipe Darcy-Weisbach friction factor (abbr. *ffp*), found in the *Property Grid* under *Pipe Results*, is $f = 0.01807$.

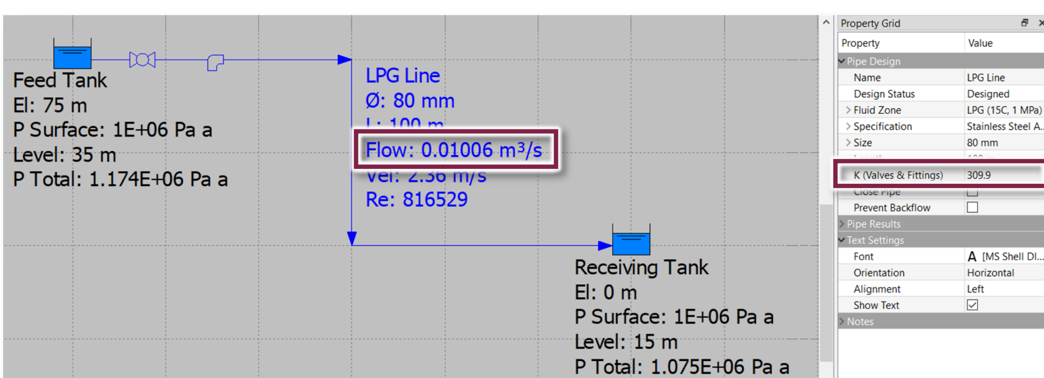


Figure 5.10: *A reduced seat diameter of 29.5 mm producing a minor loss coefficient of* $\sum K = 309.9$ *will permit a flow rate of 0.01* m^3/s.

- The pipe length, as input, is $L = 100$ m.
- The pipe hydraulic diameter (*inner pipe diameter*) is seen in the *Property Grid*, having a value of $D_h =$ 73.66 mm = 0.07366 m.
- The average velocity in the pipe (abbr. *Vel*), found on the FLO-SHEET or in the *Property Grid*, is $v_{\text{avg}} = 2.36$ m/s.

Therefore:

$$H_l = (0.01807)\frac{(100\text{ m})}{(0.07366\text{ m})} \times \frac{(2.36\text{ m/s})^2}{2 \times (9.81\text{ m/s}^2)}$$

$$\boxed{H_l = 7\text{ m}}$$

For the minor losses associated with valves and fittings, we utilize the H_{l_m} term from Equation 4.11.

- The K values for *one* of the elbows, the entrance and exit losses, and globe valve are 0.5262, 0.78, 1, and 307.1, respectively, and can be found in the *Valves and Fittings* dialog box. Alternatively, you can just use the $\sum K$ value in the K (*Valves & Fittings*) field in the *Property Grid*: $\sum K = 309.9$.
- From before, $v_{\text{avg}} = 2.36$ m/s.

Therefore:

$$H_{l_m} = \frac{(2.36\text{ m/s})^2}{2 \times (9.81\text{ m/s}^2)} \times (0.5262 \times 2 + 0.78 + 1 + 307.1)$$

$$\boxed{H_{l_m} = 88\text{ m}}$$

Calculating the value of the total losses, using Equation 4.11:

$$H_l = 7 + 88$$

$$\boxed{H_{l_T} = 95\text{ m}}$$

Comparing our hand-calculated results with the results obtained via the PIPE-FLO simulation, we find:

- For the value of the minor loss, PIPE-FLO reports the *V&F Head Loss* to be 88.03 m. This is compared to our value of 88 m. Not bad!
- For the value of the total loss over the pipe, PIPE-FLO reports the *Head Loss* as 95 m. Right on!
- PIPE-FLO does not directly report the major frictional losses, but this can be ascertained by subtracting the minor loss value from the total loss value to get 6.97 m. Close again!

The differences that we see between the hand-calculated results and PIPE-FLO's results are a direct consequence of the truncation of digits and rounding errors.

5.2 Splitting Pipes and Creating Nodes

Suppose that you need to build a continuous pipeline, but you only have enough pipe of a certain size to complete a portion of that pipeline. You have to use different size piping to complete the rest. Such a pipeline is not far-fetched. In industry, many piping systems contain *piping enlargements or contractions* for a variety of reasons: from enabling pipes to fit pre-existing equipment, to increasing the fluid velocity (and turbulence) to preventing scaling. PIPE-FLO has provisions to allow the user to join sections of dissimilar piping.

Let us take our LPG model from the previous section and modify the LPG Line to be constructed out of two different pipeline segments, with the following conditions:

- The first segment will be 25 m long; the second segment will be 75 m long.
- The first segment will incur the entrance loss, one elbow joint, and the globe valve.
- The second segment will incur the exit loss, and one elbow joint.
- The first segment will be made of the same 80 mm nominal diameter piping as before, but the second segment will only have a 50 mm nominal diameter.
- The pipe junction will occur at an elevation of 60 m above the bottom of the lower cylinder.

To meet these criteria, there are two methods:

1. Splitting an existing single pipe into two individual pipes; or
2. Creating a new pipeline with multiple segments.

We will begin by considering the first scenario.

Splitting Pipe Segments. We can split the LPG Line into two segments by selecting (left-click) the pipe, then right-clicking on it and selecting "*Split Pipe...*". The *Split Pipeline* dialog box is then opened allowing the user to specify how the pipe should be split. This process is seen in Figure 5.11. By default, Pipe-Flo splits the pipe *in half*, where each pipe segment has the same length. The first segment receives all the valves, fittings and other components. We do not want the default parameters, so make the following changes:

- Turn off *Calculation Mode*.
- Change the *Name* of the upstream pipe to be "LPG Line 1" and the downstream pipe to be "LPG Line 2".
- Set the upstream pipe to have a length of 25 m, and the downstream pipe to have a length of 75 m.
- Select the exit loss component from the *Valves and Fittings* panel, and click the [>] button to move that component to the downstream pipe. (The [>>] button will move *all* components to the downstream pipe.) If you err, use the opposite arrow to move the component back to where it was.
- The entrance loss and globe valve can stay where they are, on the default upstream side.

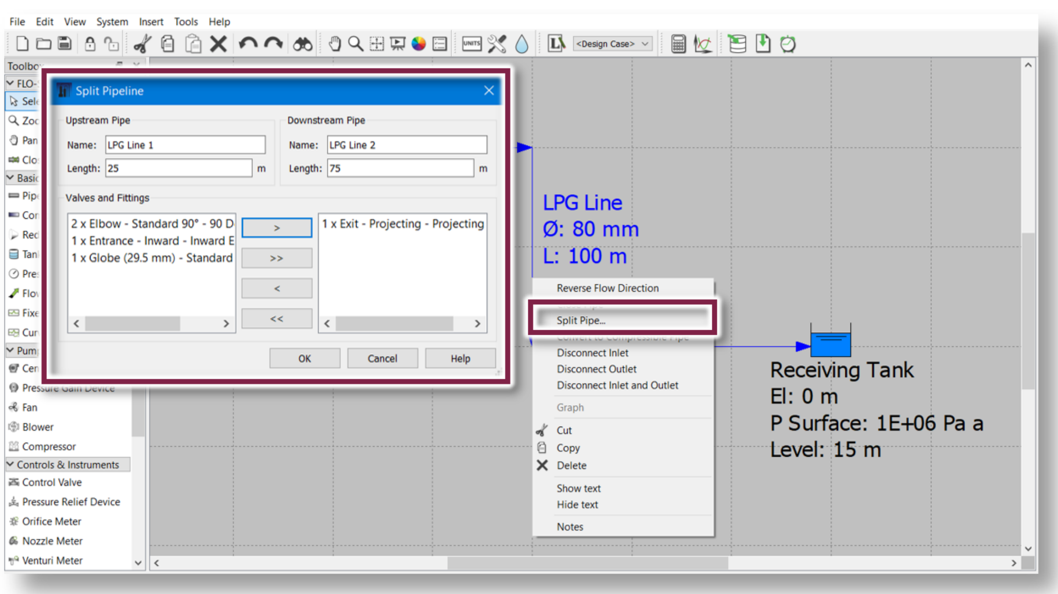


Figure 5.11: *The process for splitting a single pipe into two segments.*

If you try to move *one* of the 90° elbows over to the downstream pipe, you will notice that *both* elbows will be transferred. Herein lies a problem with splitting pipes: identical "grouped" devices cannot be separated.

To get around this issue, we will have to delete one elbow from the upstream *Valves and Fittings* dialog box and add one in the downstream dialog box.

- Leave the elbows where they are: assigned to the upstream pipe segment.
- Exit the *Split Pipeline* dialog box by clicking OK.
- Select the upstream pipe segment (prior to the purple circle icon[1] that now divides the pipe into two segments) as seen in Figure 5.12. As before, access the *Valves and Fittings* dialog box.
- Select the elbow valves from the *Installed* panel, and set the *Count* field to "1". Click Save. There is now *just one* elbow valve in the upstream pipe segment, as seen in the *Installed* panel. Click OK.
- In the same fashion, access the *Valves and Fittings* dialog box for the downstream pipe segment. Select "Elbow - Standard 90°" from under the *Fitting* heading, ensure that the *Count* field reads "1", then click Add. The elbow should appear in the *Installed* panel along with the exit loss component. Click OK.
- Re-enter the *Property Grid* for the downstream pipe section and change its *Size* to 50 mm.

All valves, fittings, and other losses are now in their correct pipe segments! Note in Figure 5.12 how the "elbow" icon can also be found on the downstream pipe segment, and how the nominal pipe diameters of the two segments differ.

Nodes. Finally, we have to deal with the *node* that was created, earlier: the purple circle at the junction between LPG Line 1 and LPG Line 2, shown in Figure 5.12. A node is essentially a bookkeeping device the PIPE-FLO has created in order to account for discontinuities in piping systems. It has no physical significance, however, node properties must be specified, and node results can also be displayed.

As before, the properties of a node as well as its simulation results can be displayed directly on the FLO-SHEET by selecting the applicable properties in the *Device View Options* dialog box.

- Click on the node to select it.

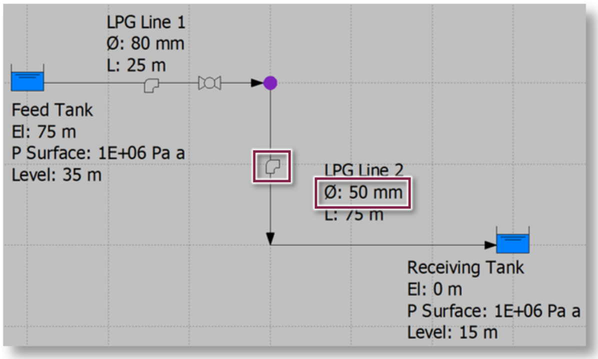


Figure 5.12: *The* FLO-SHEET *after redistributing the pipe elbows and setting the correct pipe diameters.*

[1] This is a *node*, which we will deal with momentarily. The purple colour advises the user that the design parameters have not yet been satisfied. When they are satisfied, the node will turn black.

- Enter the *Property Grid*. The node has its own *Node Design* heading, as seen in Figure 5.13.
- Set the *Name* of the node: in this example, we will call it "LPG Pipe Junction".
- Set the *Elevation* of the node. Physically, this is the height at which the pipes join, and as specified previously, this value is 60 m.

Adding a Pipe Contraction. Following the specification of the node, we will make one addition to the system. Given that we have a *pipe contraction* occurring at the location of the node, this also constitutes a minor loss! As is such, we will add a pipe contraction to the pipeline, illustrated in Figure 5.14. In this example though, we will attach the contraction on the upstream pipe—and as it turns out, it does not matter to which pipeline segment that we it add it. PIPE-FLO will move the fitting to the correct end of the pipe.

- Access the *Valves and Fittings* dialog box for the *upstream pipe*, LPG Line 1.
- Under the *Fitting* heading, select *Reducer - Contraction*.
- In the *Diameter* field, PIPE-FLO is looking for the *nominal pipe diameter that does not correspond to that pipe*. In this case, the upstream pipe is where we are adding the fitting, so we use the nominal diameter of the downstream pipe, here: 50 mm.

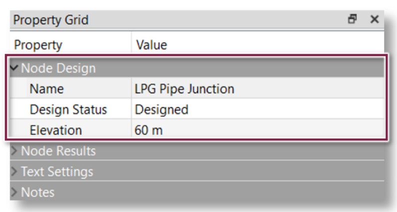


Figure 5.13: *Adding Node Design parameters in the Property Grid.*

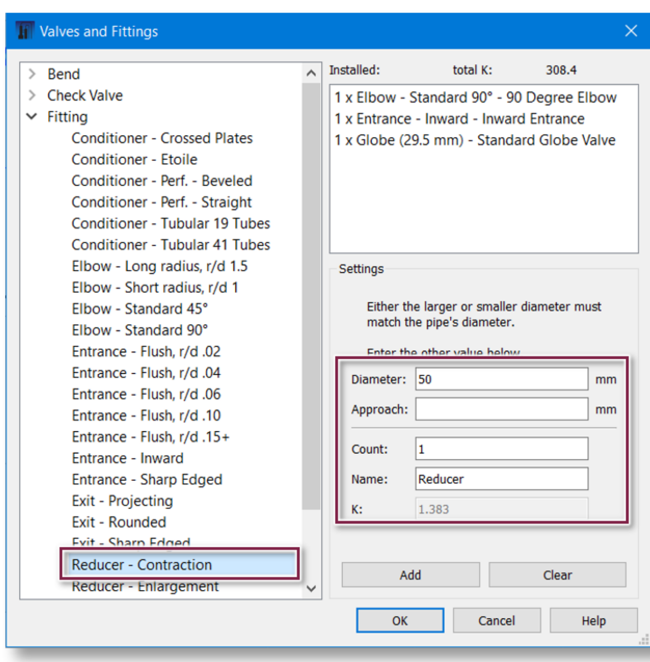


Figure 5.14: *Adding a pipe contraction to the FLO-SHEET.*

- In the *Approach* field, PIPE-FLO suggests that the user inputs the *approach length* of the pipe contraction: the length over which the pipe contracts. This example does not provide an approach length, so we will leave this field blank, which corresponds to an approach length of zero. A zero approach length means that the pipe junction occurs at a flat face: there is no gradual reduction in diameter.
- Set the *Count* field to "1", and choose a *Name* for your pipe contraction.
- Click Add; the pipe contraction will appear in the *Installed* panel.
- Click OK, exiting the dialog box.

It may be noted that *pipe expansions* can be added in a similar fashion.

We are now ready to begin simulating our system, again! Re-enter *Calculation Mode* at this time to display the results, seen in Figure 5.15. Here is a comparison of the current simulation results to the ones at the time that the globe valve was installed, and the volumetric flow rate was adjusted to 0.01 m^3/s:

- With the addition of the narrower pipe segment, the volumetric flow rate Q has decreased from 0.01006 m^3/s to 8.456×10^{-3} m^3/s.
- The average velocity of fluid v_{avg} has decreased from 2.36 m/s in LPG Line 1 to 1.984 m/s, but has increased to 4.439 m/s in LPG Line 2.
- The Reynolds Number Re has decreased from 816529 to 686460 in LPG Line 1, but has increased to 1.027×10^6 in LPG Line 2.

A critical fluid mechanics concept to realize is with the constriction in pipe diameter going from LPG Line 1 to 2, *the volumetric flow rate does not change with the pipe diameter*! However, as the pipe constricts, *the fluid's average velocity increases with the decrease in cross-sectional area.*

You may also note, here, that the node has its own set of simulation results, which can be accessed from the *Property Grid*. See Figure 5.16 for the current results from the "LPG Pipe Junction": the node pressure and hydraulic grade.

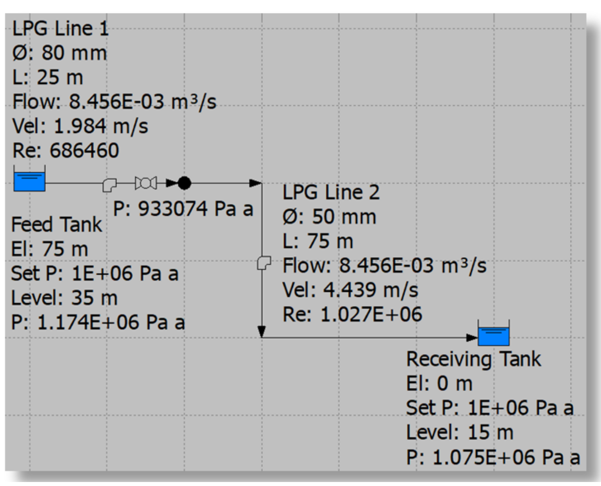


Figure 5.15: *The* FLO-SHEET, *displaying the results of all the modifications.*

At this point, save your FLO-SHEET. We will use it in the problems later in this tutorial.

Using Nodes to Segment Pipes. It was previously mentioned in this section that there are two ways to create a pipe with multiple segments and different properties per segment. The first method has been explained: divide an existing pipe by performing a *split*. While this is a good option if the pipe already exists, the second method can be used if no pipe exists that can be split, or if many segments are needed.

In this second method, the user creates a simple segmented pipeline by generating nodes and pipe segments sequentially, as seen in Figure 5.17. There is no need to split an existing pipe, or to allocate existing valves and fittings to pipe segments.

1. Start by selecting the *Pipe* tool in the *Toolbox* window.
2. Click on your starting point and *double-click* to set the location of your first node. Recall that a *single-click* creates a "bend" in the pipe.
3. Once the node has been generated, and with the *Pipe* tool still selected, click on the node to start the next segment of your pipe.
4. Repeat the second and third steps *ad nauseum* until the last node has been generated, or you have clicked on your endpoint (*e.g.* a tank), ending the pipeline.

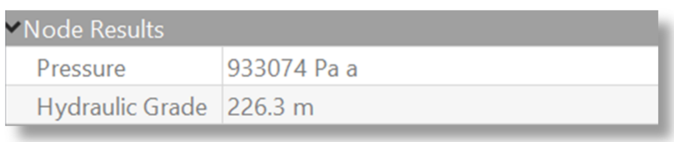


Figure 5.16: *Node results can also be found in the Property Grid.*

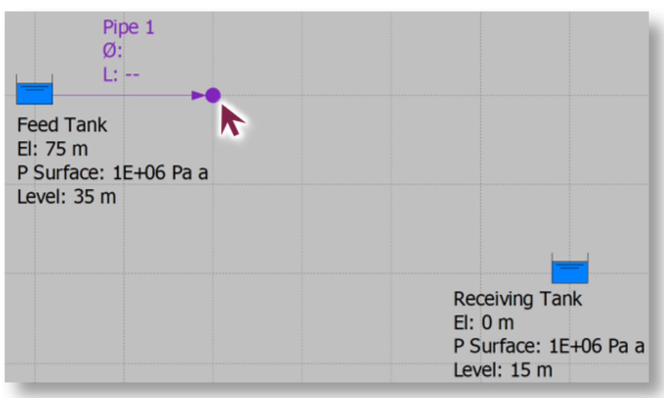


Figure 5.17: *A second method of creating a segmented pipeline, without the need to split an existing pipe.*

Tutorial Problems

Problem #1. Answer the following questions, using values from the *Valves and Fittings* dialog box. Use a pipe of your choice.[2]

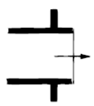
(A) What is the minor loss coefficient K associated with a standard 45° elbow fitting?

(B) What is the minor loss coefficient K associated with a standard 90° elbow fitting?

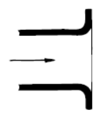
(C) What causes a greater loss (higher $\sum K$): one standard 90° elbow or two standard 45° elbows? Why do you think this is?

(D) Consider the three classes of pipe exits available to PIPE-FLO users, and seen below: (outward) *Projecting*, *Rounded* (or bell-mouthed), and *Sharp Edged*. What are their minor loss coefficients K?

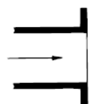
Projecting
(Outward Projecting)



Rounded
(Bell-Mouthed)

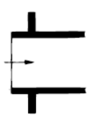


Sharp Edged
(Sharp-Edged)

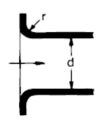


(E) Consider the three classes of pipe entrances available to PIPE-FLO users, and seen below: *Inward* (projecting), *Flush* (or rounded, bell-mouthed), and *Sharp Edged*. How do their minor loss coefficients K compare? Why do you think this is?

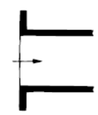
Inward
(Inward Projecting)



Flush
(Bell-Mouthed)

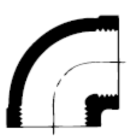


Sharp Edged
(Sharp-Edged)

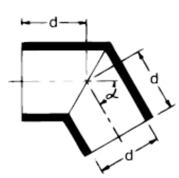


(F) A *pipe fitting* is a separate, bent component threaded onto two pipes for the purpose of joining them. Meanwhile, a *mitre bend* occurs when two pipes are directly connected (*i.e.* welded) at an angle α. By comparison, a *pipe bend* is characterized by a uninterrupted, rounded bend in the pipe with a radius r, given the pipe diameter d. Using values under the *Bend* heading, compare a 90° mitre bend with a 90° pipe bend. *Regardless of* r/d ratio, what can you say about their relative K values? Where does a 90° pipe fitting fall on this continuum of K values?

Pipe Fitting



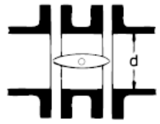
Mitre Bend



Pipe Bend

(G) What is the minor loss coefficient K of a regular (centric) butterfly valve, as seen below?

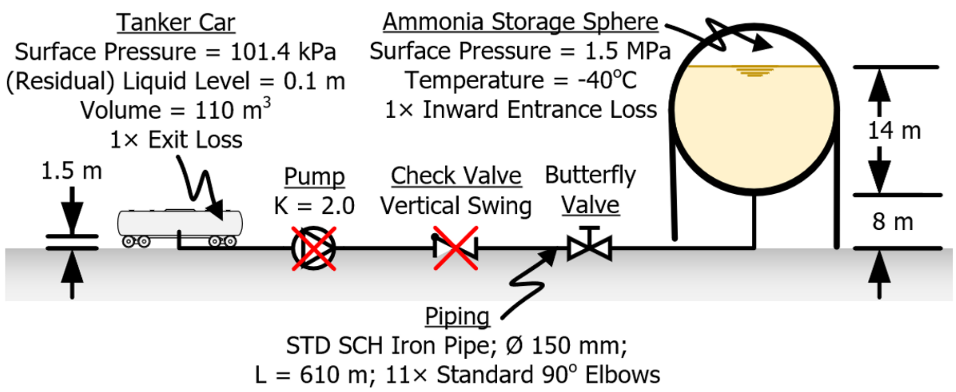
Centric Butterfly Valve



(H) What is the total minor loss coefficient $\sum K$ for a pipe with 6× standard 45° elbows, 12× standard 90° elbows, 3× 90° mitre bends, 2× regular butterfly valves, 1× rounded (flush) pipe entrance with a radius/pipe diameter (r/d) of 0.10, and 1× sharp-edged pipe exit?

[2] Pipe size/material will change the *magnitude* of your answers, but the *overall conclusions* will remain the same.

Problem #2. A manufacturer of fertilizer wishes to assess the consequences of equipment failures in its cryogenic liquid ammonia storage system, seen below. During the typical operation of this system, liquid ammonia is pumped from a tanker car into an ammonia storage sphere. Normally, a check valve is employed to prevent reversed flow in the pipe (*i.e.* in case of an emergency), however, the site engineers wish to determine the consequences if *both* the check valve and pump were to fail (*e.g.* power outage plus a stuck valve). Using PIPE-FLO, answer the following questions.



(A) Using the parameters outlined in the figure above as well as the following information, create a FLO-SHEET modeling the system.

- Create an "ammonia" *fluid zone* at -40°C and 101,325 Pa;
- Create a *Pipe Specification* for standard-schedule iron pipe;
- Use *Tank* devices to model the tanker car and storage sphere;
- Connect the tanks with a 610 meter-long iron pipe with a 150 mm nominal diameter; and
- Add the minor losses (valves & fittings) associated with the failed pump (Fixed $K = 2.0$), vertical swing-type check valve, (open) butterfly valve, and eleven standard 90°elbow fittings.

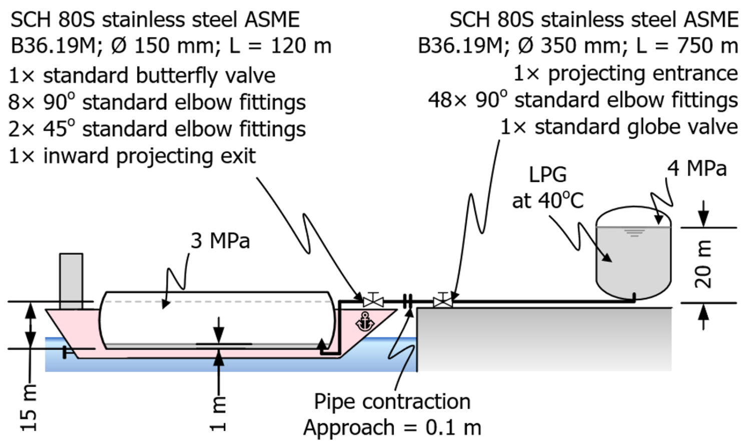
(B) Given the failure of the pump and the check valve, determine the hypothetical flow rate of liquid ammonia re-entering the tanker car.

(C) If both the pump and check valve fail, manually calculate the maximum amount of time that an operator has to react before the tanker car overflows hazardous liquid ammonia. You may assume that the storage drains at a steady rate equal to the value of Q calculated in part **(B)**.

(D) Propose and test a simple modification that could be made to make the system which would reduce the risk or severity of an accident.

Problem #3. Consider the *new* fluid statics system, below. Complete the following steps and answer the accompanying questions about this system.

Consider an unladen liquefied propane gas (LPG) tanker ship used to transport LPG overseas—similar to a standard oil tanker. Suppose that our LPG tanker is docked in Saudi Arabia, and dockside crews wish to load the ship with its cargo of LPG. Due to conditions in Saudi Arabia, the LPG temperature is 40°C with a surface pressure of 4 MPa in the dockside storage tanks. To ensure that the LPG remains in its liquefied state, the LPG tanker's storage tanks are pressurized to 3 MPa with the aid of residual LPG—as a result, there is 1 m of LPG left in the bottom of the ship's tanks. Both the tanker and dockside pipes are Schedule 80S ASME B36.19M stainless steel, however, the dockside piping has a nominal diameter of 350 mm, while the tanker's piping has only a nominal diameter of 150 mm. The lengths of the dockside and tanker piping are 750 m and 120 m, respectively. See the following figure for additional details, including a list of valves and fittings.



(A) Add a "propane" fluid zone at 40°C and 4 MPa. Then, add a "Schedule 80S ASME B36.19M stainless steel" piping specification.

(B) Add the dockside and shipboard LPG tanks.

(C) Join the tanks via the two pipe segments—dockside and shipboard—by either performing a pipe split on a single pipe segment, or by creating two pipe segments joined at a node. In any case, make sure that the node is at the same elevation as the dockside tank.

(D) Add the valves and fittings to the dockside and shipboard pipe segments according to the specifications on the diagram. Set the globe valve to be fully-open (leave the reduced seat diameter blank).

(E) Include a reducer fitting (pipe contraction) on the dock side, with a specified approach length of 0.1 m.

(F) Run your simulation. Verify that your volumetric flow rate is $Q = 0.2597$ m^3/s in both pipeline segments.

(G) Suppose that an operator wants to precisely control the volumetric flow rate using the dockside globe valve. Vary the reduced seat diameter from 10 to 330 mm, and plot a calibration curve of volumetric flow rate versus the valve's percent closure. Use the equation below:

$$\% \text{ Closed} = \frac{\text{Pipe I.D.} - \text{Reduced Seat Diameter}}{\text{Pipe I.D.}} \times 100\% \tag{5.1}$$

(H) Recall that Pipe-Flo simulates fluids systems at *steady-state*. However on inspection, you should see that as time progresses, the LPG tanker will fill with LPG, changing the dynamics of the system! (Think about what changes in the underlying Bernoulli Equation.) Assume that the pressure above the LPG remains constant (*i.e.* using a control or check valve) and the dockside storage tank is large enough that its level does not change through the process. With the dockside globe valve returned to fully-open, what is the percent difference in volumetric flow rate between when the tank is 1 m full and 15 m full of LPG? Is this significant?

End of Tutorial

Tutorial #6:
Other Devices, Valves & Meters

Tutorial #6 will consider the following concepts:

- Creating pressure boundaries and specifying flow demands
- Generic devices that cause pressure drops
- Modeling controls and instruments (*e.g.* valves and meters) that alter flow conditions
- Flow and resistance coefficients

This tutorial deals with the questions: "How do I simulate parts of a fluid system with devices not directly-specified in the PIPE-FLO software?" and "How do I compensate for the effects of controls and instrumentation?" Simply put:

- We can use the *Pressure Boundary* tool to set the *known pressure* of a fluid entering or exiting a piping system at a defined boundary.
- We can use the *Flow Demand* tool to set the *known flow rate* of a fluid entering or exiting a piping system at a defined boundary.
- We can use a *Fixed dP Device* to create a pressure drop of a *fixed size* at a point in a piping system.
- We can use a *Curve dP Device* to create a pressure drop of a *size dependent on the flow rate* at a point in the piping system.
- We can use *Controls & Instruments* to model control valves as well as flow meters.

You can view the simulated properties of all the devices discussed in this tutorial by clicking on the device and navigating to the *Property Grid* or *List View* window.

6.1 Pressure Boundaries & Flow Demands

The *Pressure Boundary* device can be used if you know the pressure at the inlet or outlet of a pipeline. Similarly, the *Flow Demand* can be used if you know the flow rate entering or exiting a system, or a portion of the system. When using these devices, no tank at the end of the pipeline is needed. This is particularly useful if you are trying to simulate a system where you receive fluid from an upstream location (or transport fluid to a downstream location), but specifics of that location are unknown.

For this tutorial, suppose that an oil refinery seeks to simulate the flow of isooctane (a component in gasoline) at 25°C from one area of the refinery to another through one of its pipes, as seen in Figure 6.1. It is known that upstream location in the refinery produces the isooctane at a flow rate of 2.3 m^3/s and at the downstream location, the required isooctane pressure is 0.25 MPa. The 1,200 metre-long pipeline is straight (with no valves and fittings) and is located on level ground. The refinery would like to know the pressure at which the isooctane needs to be supplied in order to meet the demands of the downstream facility.

Start by creating a new FLO-SHEET. Add:

- An isooctane *Fluid Zone* at 25°C and 0.25 MPa.
- A *Pipe Specification* for the Schedule 100 ASME B.36.10M steel pipeline.

We can use *Flow Demand* and *Pressure Boundary* devices to determine the pressure needed at the supply end of the pipeline. As seen in Figure 6.2, begin by adding an upstream *Flow Demand* device and then a downstream *Pressure Boundary* device to the FLO-SHEET by selecting the devices from the *Toolbox* under *Basic Devices*. Click on *Flow Demand* device and in the *Property Grid*:

- Set the *Elevation* to 0 m.
- Click on the *Operation* field which opens the *Operation* dialog box. Set the *Volumetric Flow Rate* to 2.3 m^3/s. Click OK.
- Set the *Flow Type* to *Flow In*, meaning that this *Flow Demand* is a source of *flow into the system*, as oppose to a location where *flow exits the system*.

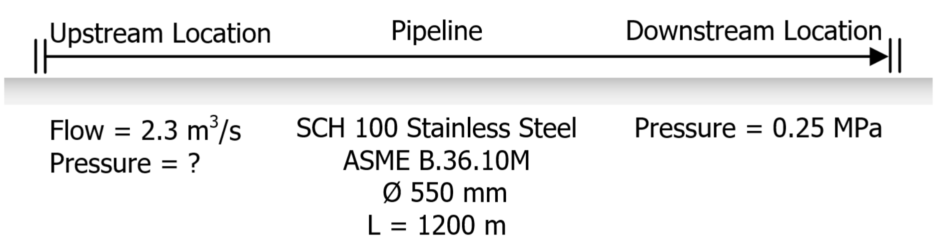


Figure 6.1: *Isooctane pipeline with a fixed supply flow rate and demand pressure.*

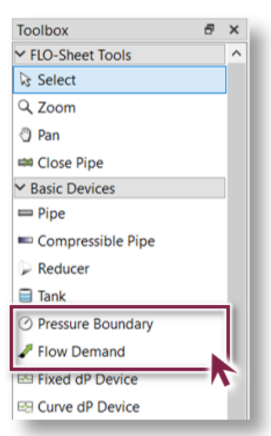


Figure 6.2: *Pressure Boundary and Flow Demand devices in the Toolbox.*

Now click on the *Pressure Boundary* device and in the *Property Grid*:

- Set the *Elevation* to 0 m.
- Set the *Operation* to 0.25 MPa.

These actions can be seen in Figure 6.3. Subsequently, connect the *Flow Demand* and *Pressure Boundary* devices with a pipe that is 1,200 m long and 550 mm in diameter. Your simulation is now ready to be run! The FLO-SHEET should appear as seen in Figure 6.4. The *Flow Demand* device reports that a pressure of 1.777 MPa is required at the supply end of the pipeline in order to ensure that 2.3 m^3/s of isooctane is received at the opposite end of the pipeline at 0.25 MPa.

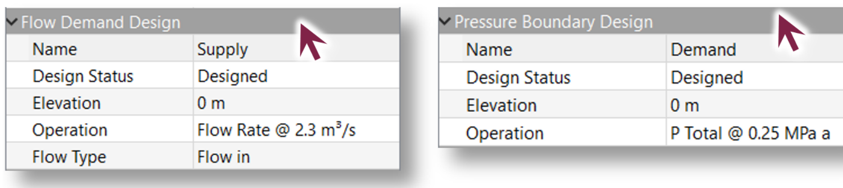


Figure 6.3: *Setting the **(L)** Flow Demand and **(R)** Pressure Boundary parameters in the Property Grid.*

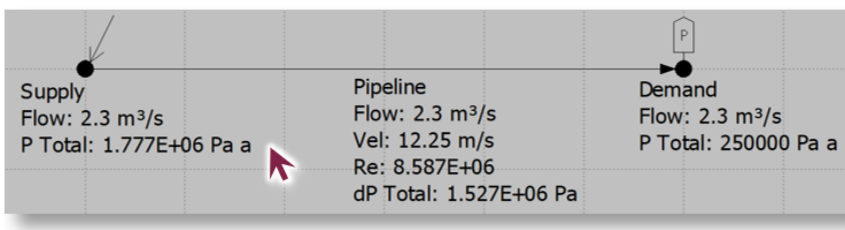


Figure 6.4: *The* FLO-SHEET *results for the isooctane pipeline simulation.*

6.2 Fixed and Curve dP Devices

A *Fixed dP Device* models a pressure drop in a piping system that is same for any fluid flow rate. While this is an *idealized scenario*, it is a useful tool if you already know the flow rate in your system, or if it is fixed. On the other hand, a *Curve dP Device* models a pressure drop in a piping system that changes with the fluid flow rate. As such, this device will ask the user for a correlation between Q and dP. These devices are useful for modeling pressure drops that occur due to process equipment not simulated by PIPE-FLO, such as columns/towers, beds, other vessels, and heat exchangers.

Now using the example from §6.1, suppose that the refinery requires that a stripping process be installed halfway down the pipeline in order to remove dissolved hydrogen sulfide gas (H_2S) from the isooctane. The refinery estimates that the relationship between the pressure drop dP due to the stripper unit and the isooctane flow rate is:

$$dP = 10^5 \times Q^2$$

Here, dP has units of Pa and Q has units of m^3/s. We will add this pressure drop to our previous model using both *Fixed dP* and *Curve dP* devices.

> Note: the removal of the H_2S has no effect on the volumetric flow rate as it was originally *dissolved* in the isooctane. We will also assume that the stripper causes no change in temperature.

To see the effects of this pressure drop, let us begin by *duplicating* the FLO-SHEET that we created in the previous section. Click and drag your cursor across the FLO-SHEET to select the entire system. Use Ctrl + C and Ctrl + V to copy and paste a duplicate of the FLO-SHEET. After the "paste" operation, click on the FLO-SHEET to select the location where this new system will be added.

PIPE-FLO will automatically rename all devices in the duplicate system to avoid having two devices of the same name.

We will add a *Fixed dP* device to our first system and a *Curve dP* device to the second system. Like before, both these devices are found in the *Toolbox* under *Basic Devices*, as seen in Figure 6.5 . To add a *dP* device, select it from the *Toolbox* and click on the pipe segment where you would like to locate the device. This causes PIPE-FLO to immediately request a *pipe split*, as explained in §5.2 on page 57. As seen in Figure 6.6, name your upstream and downstream pipeline segments and define their length: in this case, 600 m each. Click OK to finish the operation.

A *dP* device can be added directly to an existing *node*, alleviating the need to perform a pipe split operation.

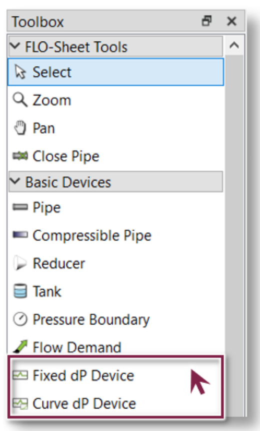


Figure 6.5: *Fixed and Curve dP Devices in the Toolbox.*

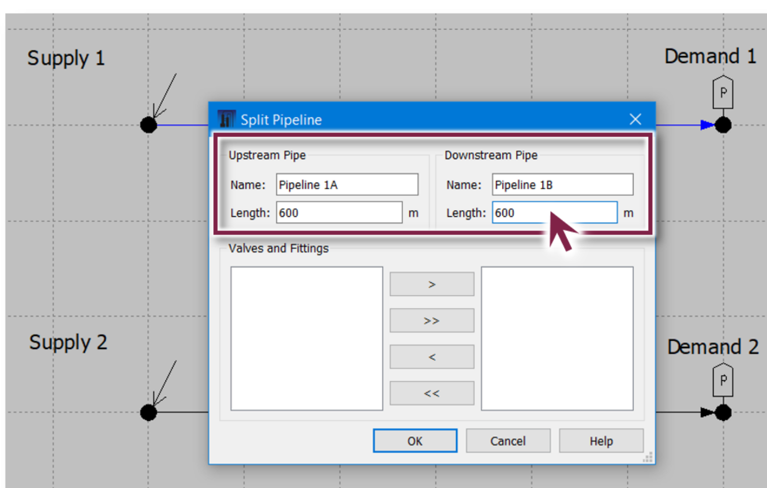


Figure 6.6: *Adding a dP device to an existing pipeline causes* PIPE-FLO *to request a pipe split.*

Once both the *Fixed dP* and *Curve dP* devices have been added to your FLO-SHEET, their parameters must be specified. Begin by clicking on your *Fixed dP* device. Under the *Fixed dP Device Design* heading in the *Property Grid* specify the following parameters, as seen in Figure 6.7:

- Set both the *Inlet Elevation* and *Outlet Elevation* to 0 m.
- Set the fixed dP to be 0.529 MPa using the Q/dP relationship on page 67 where $Q = 2.3$ m^3/s.

Subsequently, click on the *Curve dP* device and navigate to the *Curve dP Device Design* heading in the *Property Grid* as highlighted in Figure 6.8.

- Set both the *Inlet Elevation* and *Outlet Elevation* to 0 m.
- Click on the *Curve* field and the *Curve Data* dialog box will be opened. Type in a *Description* of the curve that will be specified.
- Click on the Estimate Curve Data... and the *Curve Data Estimator* dialog box will be opened as seen in Figure 6.9. In here, PIPE-FLO assumes a quadratic relationship between dP and Q and requires four valies in order to estimate the relationship. Set the *Static Pressure Drop* to be 0 Pa: it is the value of dP when $Q = 0$.
- For the *Flow Rate* and *Pressure Drop* fields, we will use a data point generated from the relation between dP and Q on page 67, as seen in Figure 6.9. Here, set the *Flow Rate* to be 0.5 m^3/s and the *Pressure Drop* to be 25000 Pa.
- For the *Maximum Flow Rate*, enter the highest value of Q that you expect to be needed. PIPE-FLO will then create a list of points between zero and this maximum value, along with the corresponding dP values. Since our system is operating at a flow rate of 2.3 m^3/s, we will use 4 m^3/s as our *Maximum Flow Rate*.

Because PIPE-FLO *interpolates* between the data points in the table, if different people choose different dP/Q points, PIPE-FLO may converge on slightly different simulation results!.

The FLO-SHEET is now complete, and we are now ready to re-run the simulation. Your FLO-SHEET should appear similar to the one presented in Figure 6.10. Notice the pressure drop values associated with the two dP devices:

- For the *Fixed dP* device, $\Delta P = 0.529$ MPa, as specified.
- For the *Curve dP* device, $\Delta P = 0.530$ MPa, as PIPE-FLO *linearly interpolated* from the data that was specified in order to estimate ΔP for $Q = 2.3$ m^3/s.

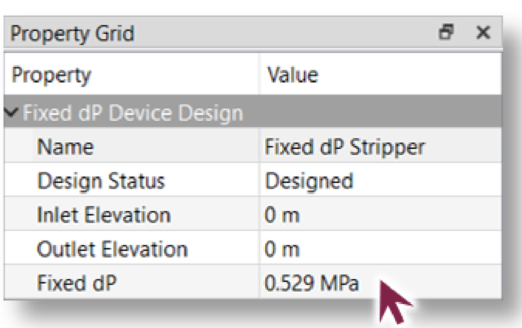


Figure 6.7: *Specifying Fixed dP Device design parameters.*

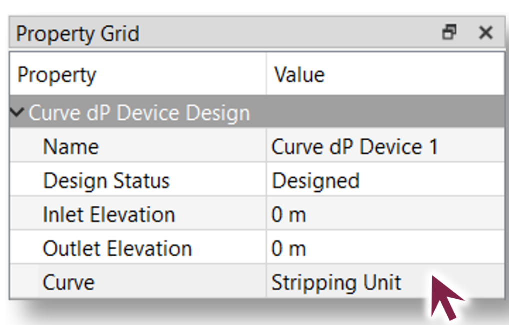


Figure 6.8: *Specifying Curve dP Device design parameters.*

Accordingly, the pressures required at the supply end of the pipeline are 2.306 MPa for the *Fixed dP* device and 2.307 MPa for the *Curve dP* device—slightly different results for the same system!

> Consider what would happen if the supply flow rate Q was varied. How would the pressure drop inherent with each dP device change?

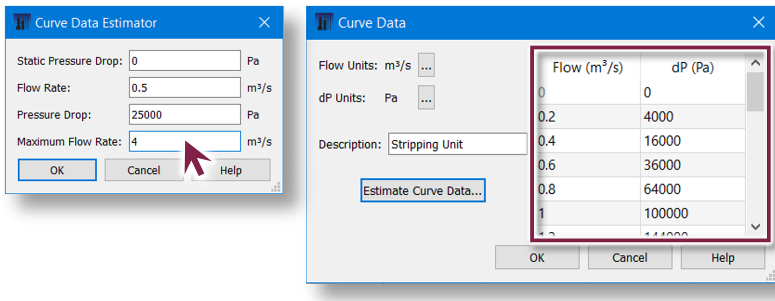


Figure 6.9: *Specifying parameters for the Curved dP Device so that* Pipe-Flo *can estimate the relationship between dP and Q.*

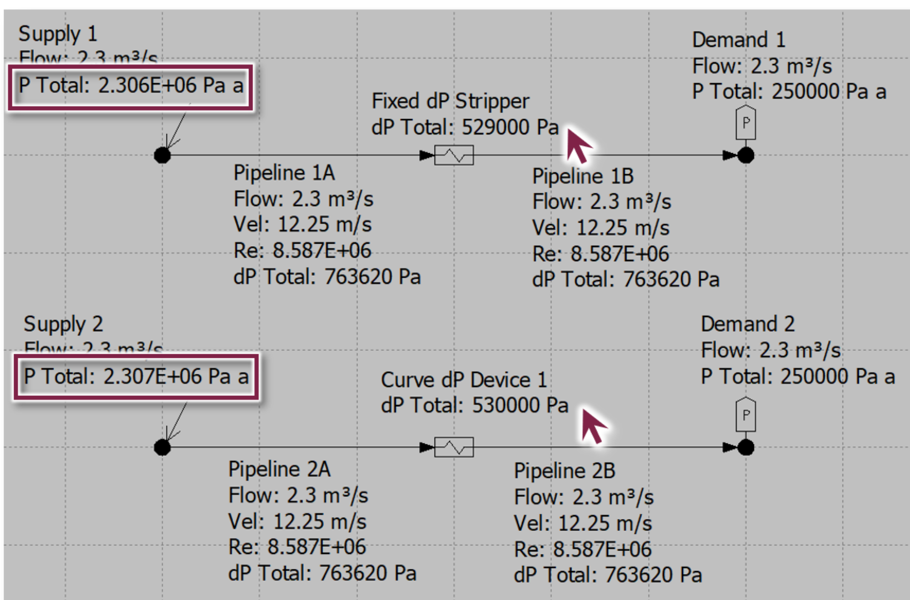


Figure 6.10: *Simulating the isooctane pipeline with a H_2S stripper modeled as both Fixed dP and Curve dP Devices.*

6.3 Controls & Instruments

Control Valves In Tutorial #5, we used valves and other components with minor losses in order to control flow conditions in our piping systems. In particular, we manipulated the *reduced seat diameter* of a valve to cause a pressure drop/a reduction in flow rate. In practice, these components with minor losses are typically seen as "fixed" and to manipulate flow conditions dynamically, *control valves* are employed, instead. According to Pipe-Flo, such valves change the geometry (*i.e.* shape and size) of the flow passage (and therefore the valve coefficient C_V), which adjusts the valve's resistance to flow. Energy from the fluid is dissipated by the valve through heat, noise, and vibration. Pipe-Flo offers a valve device that allows for both fixed and dynamic systems.

- The *Control Valve* can be used for "set-it-and-leave-it" configurations, and can also adapt the valve position in order to maintain steady state in dynamic systems requiring process control.

Let us begin by modifying our isooctane pipeline that was created in §6.1. Suppose that the refinery begins to experience fluctuating production rates upstream of the isooctane pipeline. As illustrated in Figure 6.11, the refinery installs a *swing tank* prior to pipeline so that the downstream production location will have a steady supply of isooctane. The *pressure at the tank outlet* is kept relatively constant (2 MPa) by a pressurization system while the liquid level fluctuates with production. To ensure that the downstream location receives isooctane at a steady rate of 2.3 m^3/s, a flow control valve is installed halfway down the pipeline. For this example, we will remove and ignore the stripping unit.

Begin by clicking and dragging your cursor across the Flo-Sheet to select one entire system; type Delete to remove that system from the Flo-Sheet. Remove the *Flow Demand* and *dP Device*, as well. You will notice that *nodes* (●) are left behind where the *Flow Demand* and *dP* devices once were. Then as shown in Figure 6.12, a *Control Valve* device can be selected from the *Toolbox* under *Controls & Instruments*. By clicking on the node in the middle of the pipeline, the *Control Valve* will be installed at that location without the need for performing a pipe split. Similarly, add a simple tank to the supply end of the pipeline. In the *Property Grid* for the tank:

- Set the *Elevation* to be 0 m.
- Set the *Surface Pressure* to be 2 MPa and the *Liquid Level* to be 0 m. This corresponds to a constant pressure at the bottom of the tank of 2 MPa, as prescribed.
- Set your *Fluid Zone* to be isooctane.

The properties for the *Control Valve* can then be set in the *Property Grid*, as seen in Figure 6.13.

- Set the *Elevation* to be 0 m.
- The *Valve Data* allows users to enter additional design data required for more calculations. We will leave this field blank (--).
- Click on the *Operation* field, and the *Operation* dialog box will be opened. Here, you can choose what you want your valve to do:

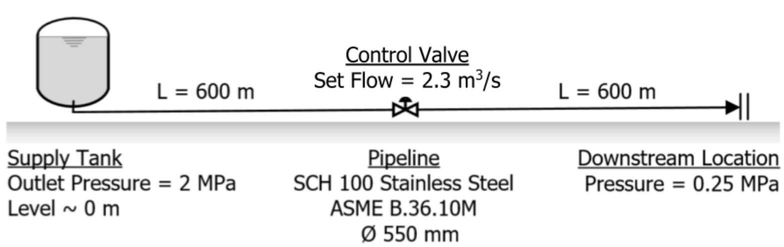


Figure 6.11: *The refinery adds a swing tank to the pipeline to smooth out production variations.*

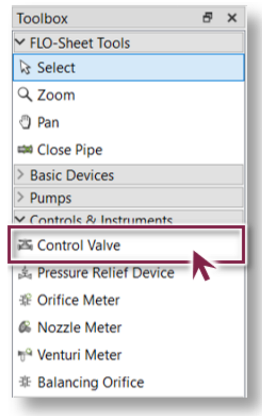


Figure 6.12: *The Control Valve device can be found in the Toolbox.*

- ▷ The *Flow Rate* option fixes the *flow rate* of fluid through the valve by manipulating the valve accordingly.
- ▷ The *Discharge Pressure* option fixes the *discharge pressure* of the valve.
- ▷ The *Suction Pressure* option fixes the *suction pressure* of the valve.
- ▷ The *Differential Pressure* option fixes the drop in pressure across the valve
- ▷ The *Fixed Flow Coefficient* option sets the *flow coefficient* for the valve.
- ▷ The *Manual Position* option is used in conjunction with the *Valve Data* field.

- Choose the *Flow Rate* option and specify the *Volumetric Flow Setting* to be 2.3 m^3/s. Click OK.
- The *Allowable dP min.* and *Allowable dP max.* fields allow the user to set the minimum and maximum pressure drops that are allowable across the device. Although these settings have no effect on the calculation, PIPE-FLO will show an error if one of these constrains is exceeded. We will leave these fields blank (--).

We are now ready to run our system, again. Figure 6.14 shows the results of the simulation. Note that in order to achieve a volumetric flow rate of 2.3 m^3/s, PIPE-FLO finds that the valve must have a C_V (flow) coefficient such that the pressure drop across the valve is 0.223 MPa.

Flow Meters. PIPE-FLO also contains provisions for different types of flow meters; such devices almost always induce a pressure drop (which is related to the flow rate), so it is useful to include them in our models. These flow meters are design such that their geometry creates a pressure drop in the system, and as learned in class, we can use the size of this pressure drop along with other parameters in the system to calculate the flow rate through the

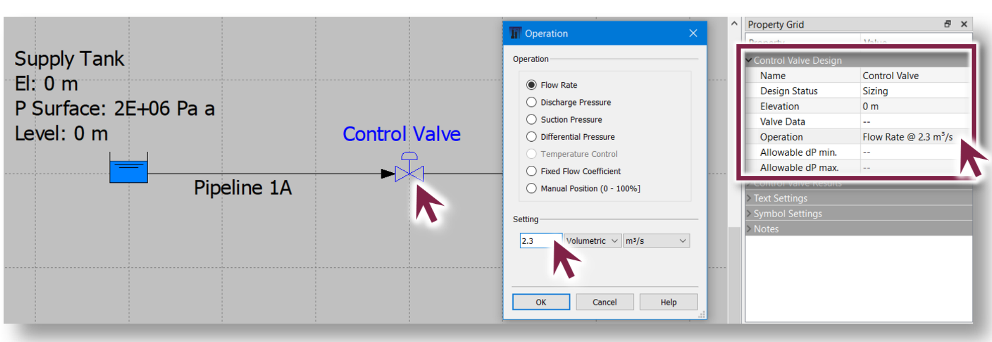


Figure 6.13: *Setting the parameters and Operation field for the Control Valve.*

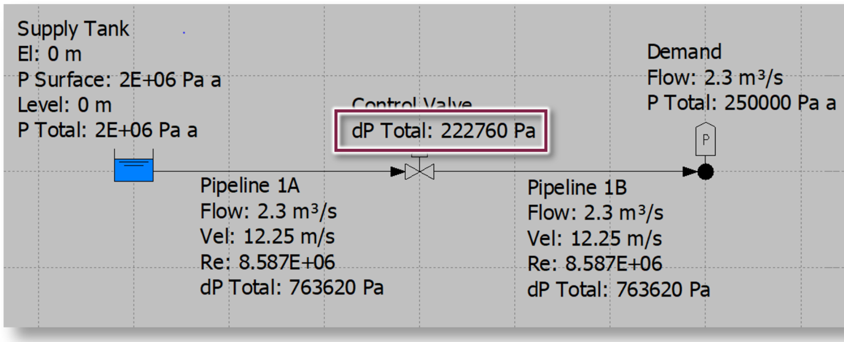


Figure 6.14: *The* FLO-SHEET *featuring results of the simulation with the Control Valve device.*

meter. In this courseware, we will not delve too far into the details of these flow meters, but be aware that they can be modeled using the PIPE-FLO software. PIPE-FLO includes three different types of flow meters:

1. The *Orifice Meter*;
2. The *Nozzle Meter*; and
3. The *Venturi Meter*.

Other types of flow meters can be roughly modeled by adding a minor loss, *Fixed dP* or *Curve dP* device to your pipe.

Using our scenario from before, let us assume that the refinery does not wish to set the flow rate of the isooctane pipeline to be 2.3 m^3/s, but instead, they want to measure the flow rate with an orifice flow meter, as seen in Figure 6.15.

As before, remove the *Control Valve* device from the FLO-SHEET. Access the flow meters in the *Toolbox* under the *Controls & Instruments* heading, as seen in Figure 6.16. Add a flow meter to the node in the center of the pipeline; in our case, we will use an *Orifice Meter*. In the *Property Grid*, set the flow meter parameters, as seen in Figure 6.17:

- For all meters, set the *Elevation*. In our case, we will use 0 m.
- Also for all meters, choose a *Device Material* (what the meter is made of) and a *Pipe Material*. The *Pipe Material* describes the material of the short conduits leading in and out of the meter to the system's piping. In our case, we will use *300 Series Stainless Steel* for both the *Device* and *Pipe*.
- For a *Orifice Meter*, choose a *Pressure Tap*: *Corner*, *1D - 1/2D*, or *Flange*. A *Pressure Tap* is like a pressure probe which is inserted into the orifice meter. In our case, keep the default *Corner* option.

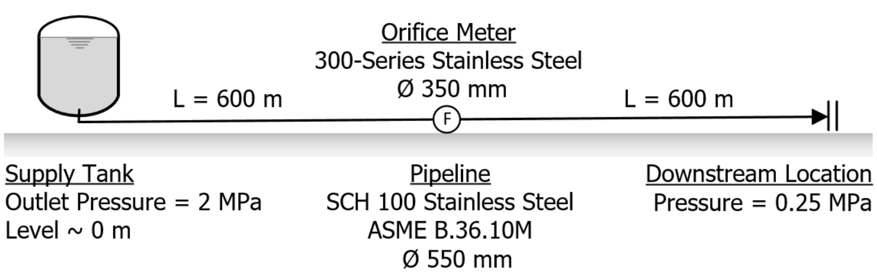


Figure 6.15: *The isooctane pipeline is fit with an orifice flow meter.*

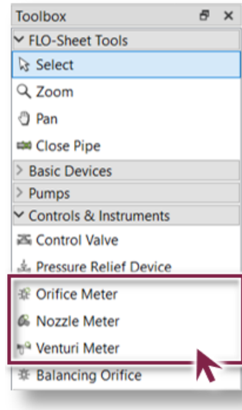


Figure 6.16: *The flow meter devices can be found in the Toolbox.*

- For a *Nozzle Meter*, choose a *Nozzle Type*: *Long Radius* or *ISA 1932*. Changing the nozzle geometry will affect the pressure drop across the meter.
- For a *Venturi Meter*, choose the *Manufacturing Method* (*As-Cast*, *Machined*, or *Rough-Welded*) and the *Divergence Angle* (*7°* or *15°*). The *Manufacturing Method* and *Divergence Angle* change the frictional losses and the degree of pressure drop across the meter, respectively.
- For all meters, set the *Diameter* to be the diameter of the interior of the flow meter. In our case, we will use 350 mm.

After all the parameters have been specified, the FLO-SHEET can be simulated. Your FLO-SHEET should appear similar to the one shown in Figure 6.18. Observe that the flow rate of isooctane in the pipeline is 2.314 m^3/s—remarkably close to the original 2.3 m^3/s, given the outlet pressure of the isooctane tank. From the results, it is apparent that the *Orifice Meter* creates a pressure drop of approximately 0.204 MPa.

When we consider Venturi meters in class to relate pressure drop and flow rate, the pressures are measured at the location of the pressure taps. As such, the *Pressure Taps dP* is the value that you would use to calculate the flow rate through the meter if you were to perform this calculation by hand. However, the *Total Pressure Drop* refers to the loss of pressure over the entire meter. Recall that we have short conduits that connect the system piping and the meter. The *Total Pressure Drop* includes these conduits.

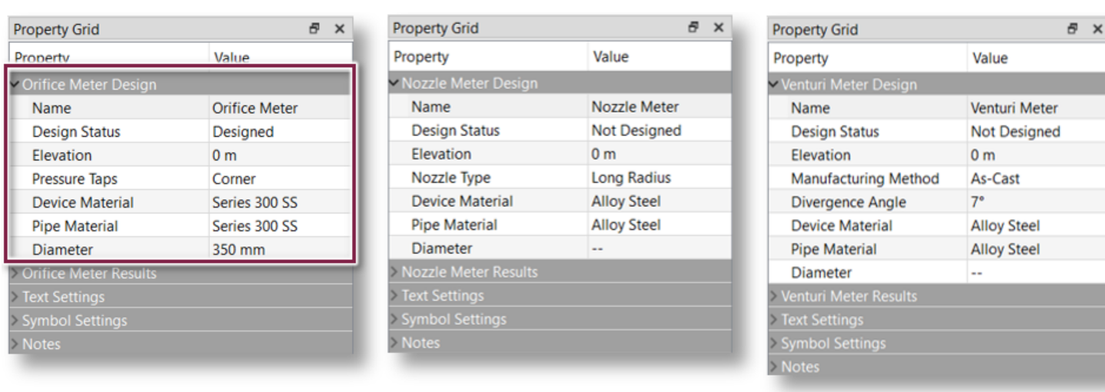


Figure 6.17: *Setting the parameters of* **(L)** *orifice,* **(C)** *Nozzle, and* **(R)** *Venturi flow meters in the Property Grid.*

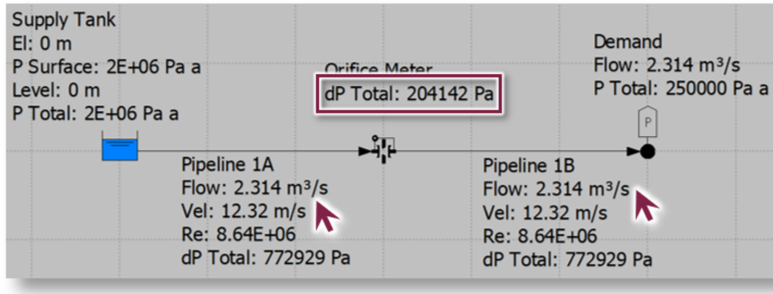


Figure 6.18: *Simulation results for the case where the refinery installs an orifice flow meter on its isooctane pipe.*

6.4 Resistance and Flow Coefficients

The *flow coefficient* (C_V or K_V) is a unitless number that represents how much flow capacity that an obstruction allows. On the contrary, a *resistance coefficient* K is a unitless number that quantifies how much resistance an obstruction creates. According to PIPE-FLO, C_V is defined as the flow rate (in gpm) of 60°F water which causes a differential pressure of 1 psi when passed through a valve. Similarly, K_V is defined as the flow of water in m^3/h (where $\rho = 1000$ m^3/h and $\mu = 0.001$ Pa·s) which causes a differential pressure of 1 bar when passed through a valve. These values are important in systems design; manufacturers of control valves, fittings, and other piping devices provide at least one of these values to customers so that they know the effect of the device in their piping system. Whereas most fluid mechanics textbooks use K, valve and fitting manufacturers more commonly employ C_V/K_V, where the former is in US units, and the latter is in SI units. Furthermore, you can convert between C_V and K_V via Equation 6.1:

$$K_V = 0.865 \times C_V \tag{6.1}$$

Empirically, C_V can be calculated via Equation 6.2, where γ is the fluid's specific gravity. Note that this equation is in imperial units, since C_V is *not SI*:

$$C_V = \frac{Q}{\sqrt{\Delta P/\gamma}} \tag{6.2}$$

Similarly, C_V/K_V are related to K via the Equation 6.3, where d is the pipe diameter in *inches*. We can relate the value of C_V/K_V, given to us by the manufacturer, to the K value we use in this course. Intuitively, C_V/K_V and K are inversely related: higher values of C_V produce lower values of K.

$$K = 890.9 \times \frac{d^4}{{C_V}^2} \tag{6.3}$$

ℹ Where have you seen K before? As it turns out, the *resistance coefficient* is the same K as used when calculating minor losses!

As always, you can find the values of C_V or K_V for a device by clicking on it and navigating to the *List View* window or the *Property Grid*.

Tutorial Problems

Problem #1. A stream of methanol (MeOH) at 5°C and 800 kPa is transported down a pipeline to an elevated storage sphere at a height of 12 m above the ground. The surface pressure and liquid level in the sphere are kept constant at 120 kPa and 5 m. A control valve installed 250 m down the level pipeline and a total distance of 20 m away from the bottom of the sphere is to be used to maintain a MeOH volumetric flow rate of 0.15 m^3/s. See Figure 6.19 for additional process details.

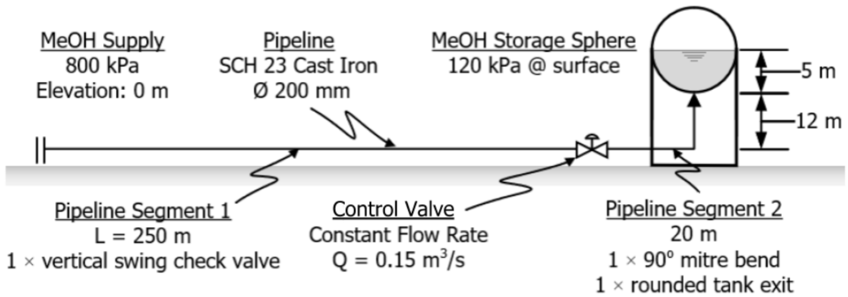


Figure 6.19: *A methanol pipeline feeds a storage sphere at a constant volumetric flow rate of 0.15 m^3/s with the help of a control valve.*

(A) Build the system described above. What is the total volumetric flow rate, and the pressure drop associated with the control valve?

(B) What is the flow coefficient C_V of the control valve? Also report this value as K_V.

(C) Record head loss H_l across the control valve and the average fluid velocity v_{avg}.

(D) From Part **(C)**, calculate the resistance coefficient K.

(E) Delete the control valve from your FLO-SHEET. Leave the node as-is. On the upstream pipe segment, add a component with an additional minor loss: in the *Valves & Fittings* dialog box under the *Other* heading, select "*Fixed K*" and set the value equal to K from Part **(D)**.

(F) Re-run your FLO-SHEET. How do the new simulation results compare to your original results from Parts **(A-C)**?

Save your FLO-SHEET from **Problem #1** as a new file. We use it as a starting point for **Problem #2**.

Problem #2. A modification is made to the system described in **Problem #1**. The supply of MeOH is changed to have a constant volumetric flow rate of 0.2 m^3/s and the control valve (elevation = 0 m) is replaced with a heat exchanger unit having a *fixed* pressure drop of 150 kPa. You may assume no changes in fluid density or viscosity.

Before you start, be sure to remove the *Fixed K* device that you added in **Problem #1E** from your upstream pipe segment!

(A) Build the system described above using your FLO-SHEET from the last problem as a starting point. Tip: be sure to set the *Flow Type* of your *Flow Demand* device to be "*Flow In*". Also, use a *Fixed dP Device*. What is the MeOH supply pressure required for this system to operate?

(B) Create a graph of the volumetric flow rate Q vs. the required supply pressure P from $Q = 0.1$ m^3/s to $Q = 0.5$ m^3/s. How does P change with Q when you use a *Fixed dP* device?

(C) Suppose that new information comes to light suggesting that the relationship between Q and dP in the heat exchanger is not fixed, but in fact follows the relationship:

$$dP = (8 \times 10^5)(1 - \exp(-Q))$$

where dP has units of Pa and Q has units of m^3/s. Create a table of nine dP/Q pairs in the range of $Q = 0.1$ m^3/s to $Q = 0.5$ m^3/s. Recall that the *Curve Data Estimator* assumes the relationship is quadratic, so it cannot be used here. Replace the *Fixed dP* device with a *Curve dP* device. In the *Curve Data* dialog box, enter your nine dP/Q pairs and re-run the simulation.

(D) On your graph from Part **(B)**, plot Q vs. P from $Q = 0.1$ m^3/s to $Q = 0.5$ m^3/s. How does P change with Q for a *Curve dP* device? Compare your results to those from Part **(B)**.

End of Tutorial

Tutorial #7:
Introduction to Pumps

Tutorial #7 will consider the following concepts:

- Using generic pressure gain devices
- The relationship between net positive suction head and cavitation
- Calculating the required net positive suction head using a centrifugal pump device
- Using gradient colors to visualize changes in flow conditions

In this tutorial, we will investigate for the first time systems where fluid transport is not only achieved by hydrostatic pressure, but also by providing additional energy to the system. *Pressure gain devices* such as pumps are used to impart head to a system, especially in cases where there is not enough energy to overcome the existing hydrostatic forces and losses.

7.1 Pressure Gain Devices

Suppose that a water treatment plant produces water (15°C, 101.325 kPa) to be delivered to a water tower which supplies a small town. Let us model this scenario, illustrated in Figure 7.1. Note that the water treatment plant contains an unpressurized holding tank which is used to feed the water tower. Practically, this means that you can set the tank surface pressure to be 101.325 kPa and its liquid level to be 0 m, given that the inlet to the pump tank is at the surface of the tank.

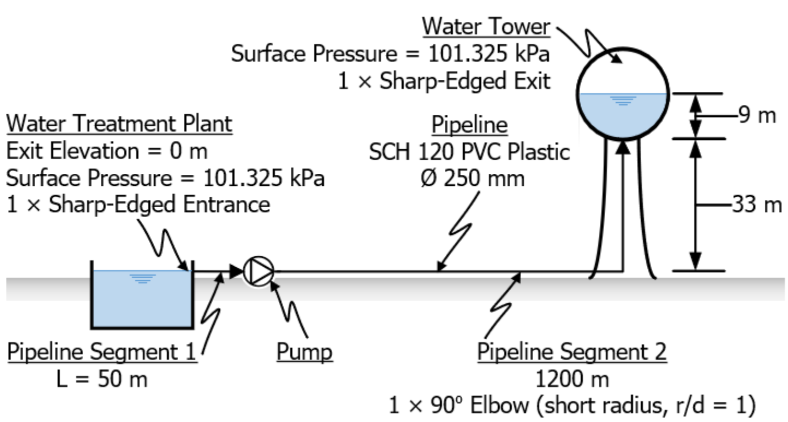


Figure 7.1: *A water treatment plant feeds a water tower.*

Begin by creating the system outlined in Figure 7.1. However, *do not add the pump, yet.* Instead, create a *node* where the pump should be. Remember to specify an appropriate elevation for the node!

Once your system has been created, run the simulation. What happens? As see in Figure 7.2, notice that the pipes are now yellow (with arrows pointing towards the treatment plant) and two Level 3 errors appear in the *Messages* window indicating “reversed flow”. For the water to flow *into* the water tower, we need a pressure gain device like a pump in order to add energy into the system to overcome the elevation change. Using the Bernoulli Equation, PIPE-FLO (or you) can calculate via Equations 7.1 and 7.2 how much pump head H_P (m) or what pressure increase ΔP_{pump} (m) is needed in order to overcome such an elevation change.[1]

$$\frac{P_1}{\rho g} + \alpha_1 \frac{v_1^2}{2g} + z_1 + H_P = \frac{P_2}{\rho g} + \alpha_2 \frac{v_2^2}{2g} + z_2 + H_{l_T} \tag{7.1}$$

$$H_P = \frac{\Delta P_{\text{pump}}}{\rho g} \tag{7.2}$$

Suppose that we already have information about a specific pump that is to be used in this system. Let us proceed by adding PIPE-FLO’s generic *Pressure Gain Device* to the FLO-SHEET. You can find the *Pressure Gain Device* under the *Pumps* heading in the *Toolbox*, as seen in Figure 7.3. Click on the device, then click on the node on the FLO-SHEET; this will place the pump directly on the node without the need for a pipe split. As seen in the *Property Grid*:

- Set the *Inlet* and *Outlet Elevations* to be 0 m as there is no elevation change within the pump, itself.
- Prior to changing any settings, the *Curve* field should be blank (--). Click on the field, and the *Curve Data* dialog box will open.

Where have you seen this dialog box before?

Here, the user enters data points that belong to the *pump curve*: at a volumetric flow rate Q, the corresponding pressure increase due to the pump is dP. In this example for *simplicity*, we will enter just two (Q, dP) points: (0 m^3/s, 0 MPa) and (0.1 m^3/s, 1 MPa). This can be seen in Figure 7.4. Add a *Description*, and click OK.

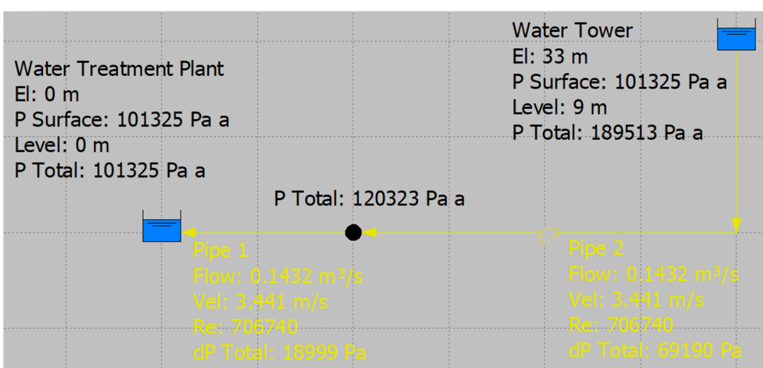


Figure 7.2: *Without a pump installed, the water flows from the higher elevation of the water tank, back to the treatment plant.*

[1] The pressure change across the pump is referred to as ΔP_{pump} in your text. PIPE-FLO refers to this value as dP. Be aware the the two nomenclatures will be used interchangeably, here.

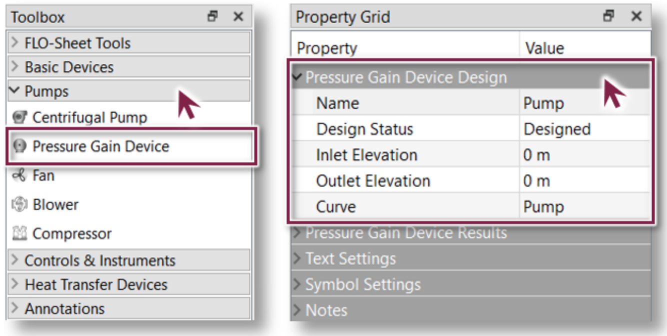


Figure 7.3: *The Pressure Gain Device can be found in the Toolbox **(L)**. As usual, navigate to the Property Grid **(R)** to edit its settings.*

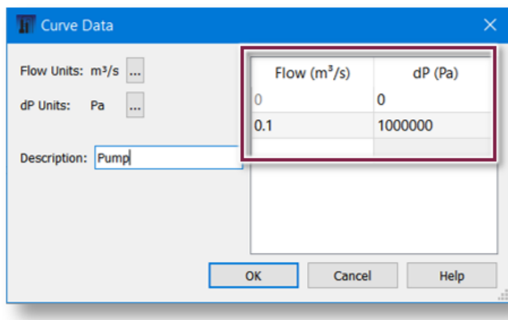


Figure 7.4: *Enter corresponding Q/dP pairs in the Curve Data dialog box.*

Figure 7.4 gives the impression that dP increases with increasing Q. As it turns out, this is generally a false assumption! As Q increases, the ability of most pumps to provide a pressure change dP decreases; thus, the pressure change produced by the pump should *decline* with increasing Q.

Having fully specified the *Pressure Gain Device*, re-run the simulation. The results are shown in Figure 7.5. Note that the pressure change across the pump—the required pressure increase for the system to operate as specified—is given as 464,215 Pa, and the volumetric flow rate is 0.04642 m^3/s. Using Equation 7.2, we can calculate that the total head generated by the pump is:

$$H_P = \frac{464215 \text{ Pa}}{(999.2 \text{ kg/m}^3)(9.81 \text{ m/s}^2)}$$

$$\boxed{H_P = 47.4 \text{ m}}$$

This is more than enough to overcome the elevation change. Unsurprisingly, our calculation matches the value for the *Total Head* recorded under the *Pressure Gain Device Results* heading in *Property Grid* or *List View* window: 47.38 m. These results can be seen in Figure 7.6.

One important disambiguation that needs to be made involves the pressure changes reported by PIPE-FLO. Recorded as "*dP Total*" on the FLO-SHEET, the *magnitude* of these dP values changes depending on the context:

- For devices like pipes and valves, it is intuitive that there should be a *pressure drop*, and thus dP should be negative. However, PIPE-FLO always expresses the *magnitude of the pressure drop*, thus dP will be shown as positive.
- For devices like pumps, there should be a *pressure increase*, and thus the value of dP is also always positive.

Therefore, keep in mind that a positive dP refers to a pressure gain with reference to a pump, but a pressure drop with reference to a pipe/valve/etc.

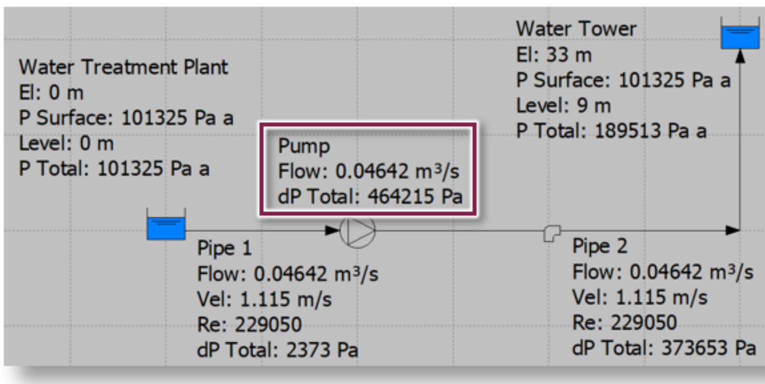


Figure 7.5: *Simulation results for the system with the water tower fed by a generic pump.*

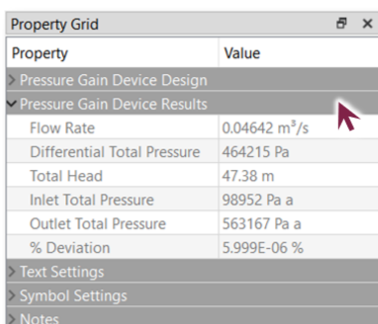


Figure 7.6: PIPE-FLO *lists the Pressure Gain Device results in the Property Grid.*

7.2 Cavitation & Centrifugal Pumps

Cavitation. *Cavitation* is the phenomenon where the local pressure in a piping system falls below the *vapour pressure* of the liquid at that temperature. As a result, a portion of the liquid spontaneously flashes into the vapour phase, which significantly disrupts the flow conditions. When these vapour bubbles collapse (and return to the liquid phase), pitting, erosion and fatigue can occur on the surface of pipes, valves, pumps, and other process equipment. In particular, "cavitation erosion" can cause the catastrophic failure of seals on pump drive shafts which are particularly susceptible to the effects of cavitation. As a result, pump and turbine manufacturers list the *Net Positive Suction Head required* ($NPSH_R$) at a pump inlet to prevent cavitation. Meanwhile, the *Net Positive Suction Head available*

($NPSH_A$) is the actual system head at the pump inlet. In order to prevent cavitation, the following condition must be met:

$$NPSH_A > NPSH_R \tag{7.3}$$

As per your course notes, the $NPSH_A$ of a pump can be calculated as:

$$NPSH_A = \frac{P_1}{\rho g} - \underbrace{(z_{\text{pump}} - z_1)}_{z_0} - H_{l_T} - \frac{P_{vp}}{\rho g} \tag{7.4}$$

where z_{pump} is the elevation of the pump inlet, z_1 and P_1 are the elevation and pressure at some *known* location upstream of the pump (often the surface of the tank which feeds the pump), H_{l_T} is the sum of major and minor losses in the *pipe connecting the tank and pump*, and P_{vp} is the *vapour pressure* of the fluid. In particular, P_{vp} can be determined from under the *Properties* heading of the *Fluid Zones* dialog box.

Centrifugal Pumps The *Centrifugal Pump* can be used to calculate the total head required to operate a piping system. In this tutorial, we will be using the centrifugal pump as a positive displacement pump by setting a constant flow rate that is required for our system. In Tutorial #8, we will consider the intricate details of the operation of centrifugal pumps.

Let us modify the water tower system from §7.1 of this tutorial. Suppose that we do not know what pump should be used for this system; we only know that a volumetric flow rate of 0.1 m^3/s is needed at the entrance to the water tower. This is an excellent opportunity to use the *Centrifugal Pump* device.

Begin by deleting the generic *Pressure Gain Device* from your FLO-SHEET from §7.1 of this tutorial. Notice that the node is left behind.

As seen in Figure 7.7, the *Centrifugal Pump* device is found in the *Toolbox* under the *Pumps* heading. Select this tool and click on the node to add a *Centrifugal Pump* to this location on the FLO-SHEET. As with other devices, click on the *Centrifugal Pump* and navigate to the *Property Grid* to set the design parameters:

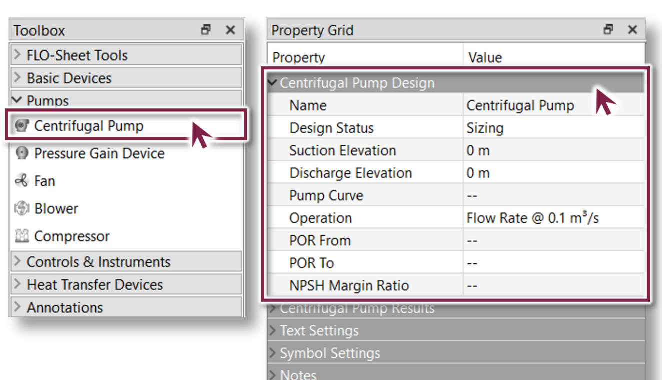


Figure 7.7: *The Centrifugal Pump device can be found in the Toolbox* ***(L)****. As usual, navigate to the Property Grid* ***(R)*** *to edit its settings.*

- Set the *Name* of the pump.
- Set the *Suction Elevation* and *Discharge Elevation* to be equal to 0 m. There is a negligible height difference across the pump from inlet to outlet.
- For now, ignore the *Pump Curve* field. We will explain and use this feature in Tutorial #8.
- Click on the *Operation* field (--); the *Operation* dialog box will be shown, as seen in Figure 7.8. Pipe-Flo allows you to choose the parameter which you would like to size the pump for:
  - *Fixed Speed*: The pump must rotate at a given speed, this feature is used when the *Pump Curve* field is specified.
  - *Flow Rate*: The pump must move the fluid at the given input volumetric flow rate.
  - *Discharge Pressure*: The pump must be sized such that its outlet pressure is the specified value.
  - *Suction Pressure*: The pump must be sized such that its inlet pressure is the specified value.
  - *Differential Pressure*: The pump must have a certain pressure drop across it.

  Here, we will set the *Flow Rate* to equal 0.1 m^3/s. Click OK.

Optionally, the user may specify:

- The minimum bound of the preferred operating region ("*POR from*"). The lower limit of the POR is a percentage of the pump's *Best Efficiency Point* (BEP) flow rate. A warning message will be shown if the flow rate through the pump falls *below* this lower limit;
- The maximum bound of the preferred operating region ("*POR to*"). Another warning message will be shown if the flow rate through the pump *exceeds* this upper limit.
- The *NPSH Margin Ratio*: the minimum acceptable ratio of $NPSH_A$ to $NPSH_R$. Another warning message will be shown if $NPSH_A/NPSH_R$ falls *below* this value.

We are ready to re-run the simulation. The results are shown in Figure 7.9. In the *Property Grid* (or directly on the Flo-Sheet), Pipe-Flo tells us that with the "operational flow" fixed:

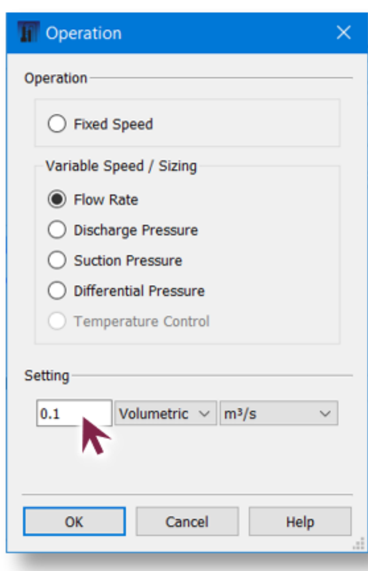


Figure 7.8: *The Operation dialog box allows the user to choose the condition for which the pump is to be sized.*

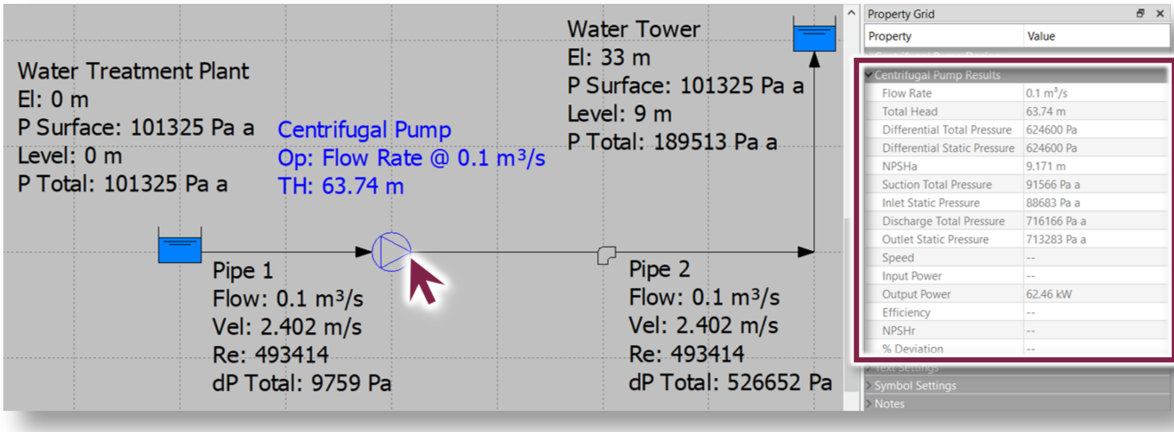


Figure 7.9: *The results of simulating the system with the water tower and Centrifugal Pump device.*

- The *Total Head* (*TH*) added by the pump is 63.74 m;
- The pump adds a *Differential Total Pressure* (*dP Total;*) of 624,600 Pa;
- The available NPSH ($NPSH_A$) at the pump inlet is 9.171 m;
- The *Suction Total Pressure* (Pipe-Flo's abbreviation: *P suct Total*)—the pressure at the pump inlet—is 91,566 Pa;
- The *Discharge Total Pressure* (Pipe-Flo's abbreviation: *P disch Total*)—the pressure at the pump outlet—is 716,166 Pa; and
- The *Flow Rate* (abbr. *Flow*) through the pump is 0.1 m^3/s.

Let us quickly verify the calculation of the head generated by this pump. In order to generate the steady state flow rate of 0.1 m^3/s, it must hold true that:

$$\text{Total Head Gained} = \text{Total Head Lost} \tag{7.5a}$$

$$\text{Pump Head} = \begin{cases} \text{Major and Minor Losses} \\ + \text{ Head Losses due to Other Devices} \\ + \text{ Head Losses due to Elevation Changes} \end{cases} \tag{7.5b}$$

Using Equation 7.5b, we will calculate the total pump head:

- By clicking on the pipes and navigating to *Pipe Results* in the *Property Grid*, we can find the total head lost (*Head Loss*) due to major and minor components. See §4.3 on page 45 for more information. For the first pipe segment, this value is 0.996 m and in the second segment, it is 20.75 m.
- In this simple system, we do not have any "other devices" that impart head losses like heat exchangers, control valves, or other process equipment, so the value of this loss is 0 m.
- The total head losses due to elevation changes is simply the difference in elevation of the fluid in the first and second tanks. From setting up this Flo-Sheet, we know that the liquid level in the Treatment Plant is 0 m and its elevation is 0 m. Similarly, the liquid level in the Water Tower is 9 m, whereas its elevation is 33 m for a total of $(33 + 9) - (0 + 0) = 42$ m of head. Thus:

$$\text{Pump Head} = (0.996 + 20.75 \text{ m}) + (0 \text{ m}) + (42\text{m})$$

$$\boxed{\text{Pump Head} = 63.75 \text{ m}}$$

Our calculation of $H_P = 63.75$ m is almost identical to Pipe-Flo's value of $H_P = 63.74$ m.

Positive Displacement Pumps. In this edition of Pipe-Flo, there is no explicit tool for modeling *positive displacement* (PD) pumps. Instead, a *Centrifugal Pump* with a constant flow rate, or a *Pressure Gain Device* can be used.

- When using a *Centrifugal Pump*, simply set the volumetric flow rate to the desired operating level in the *Operation* dialog box.
- As PD pumps can produce a variety of different heads (pressure changes) over a relatively narrow window of flow rates, when using a *Pressure Gain Device*, the pump head (pressure change) can be modeled as a *linear function of flow rate*, valid only within that narrow window. This relationship must be known *a priori* (*e.g.* from the manufacturer), and can be input in the *Curve Data* dialog box.

Gradient Colors. We can use the *Gradient Colors* tool to help visualize the flow conditions in our piping system. For example, let us use color to visualize the changes in fluid pressure in our water tower example, as seen in Figure 7.10:

- Access the *Gradient Colors* window by navigating to View > Gradient Colors or by typing Ctrl + 6. This window is often docked right below the *Property Grid*.
- In the *Gradient Colors* window, select the *Criteria* which you would like to visualize from the drop-down menu. In this case, we will select *Pressure*.
- In the *Minimum* and *Maximum Range* fields, you can (optionally) choose the lowest and highest pressure values for which the color gradient will be based upon. In this case,“90,000 kPa” and “725,000” kPa have been chosen as the minimum and maximum bounds—slightly above and below the actual range of pressure values in this system.
- Optionally, choose your *Colors* by clicking on the square icons and choosing colors from the palettes. Here, “pink” has been chosen as the “high pressure” color, “dark blue” has been chosen as the “medium pressure” color, and “light blue” has been chosen as the “low pressure” color. Pipe-Flo will interpolate between these colors, creating a gradient.

If the user does not manually choose the *Range* and *Colors*, Pipe-Flo will do this automatically with the values at-hand. As seen in Figure 7.10, it can be clearly seen that the water pressure is relatively low and stable until the pump. The pump then significantly pressurizes the fluid, and the pressure gradually decreases (due to major and minors losses and the increase in elevation) until the water tower.

Be sure to save your work, here. We will use this completed Flo-Sheet as a starting point for the next tutorial!

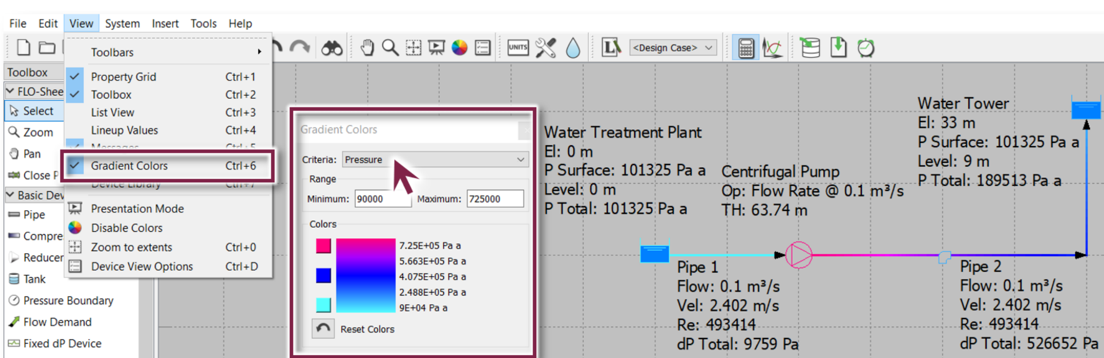


Figure 7.10: *Using the Gradient Colors tool to visualize pressure changes in the simulation with the water tower. Note that the Gradient Colors window has been undocked, here.*

Tutorial Problems

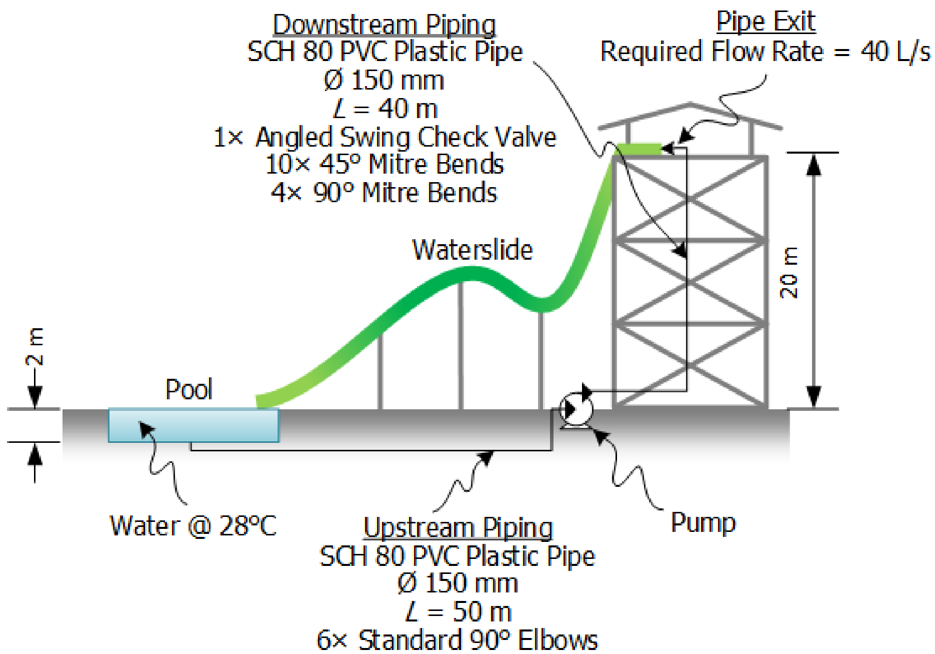
Problem #1.

(A) Conceptually, what is the difference between the *total head* produced by a pump and the pump's $NPSH_A$?

(B) If you are told that a pump produces 10 m of head, what does this mean?

(C) If you are told that a pump has an $NPSH_A$ of 10 m, what does this mean?

Problem #2. As seen in the figure below, a pump draws water stored in a swimming pool up to the top of a water slide, located 20 m above ground level. The water at the water park is kept at 28°C and 101.325 kPa. The water slide draws water through a PVC Plastic Pipe from the bottom of the pool with a water depth of 2 m. In order for riders not to get stuck, the flow rate at the entrance of the water slide must be kept at at least 40 L/s. Assume that the pump and pool both have an elevation of 0 m.

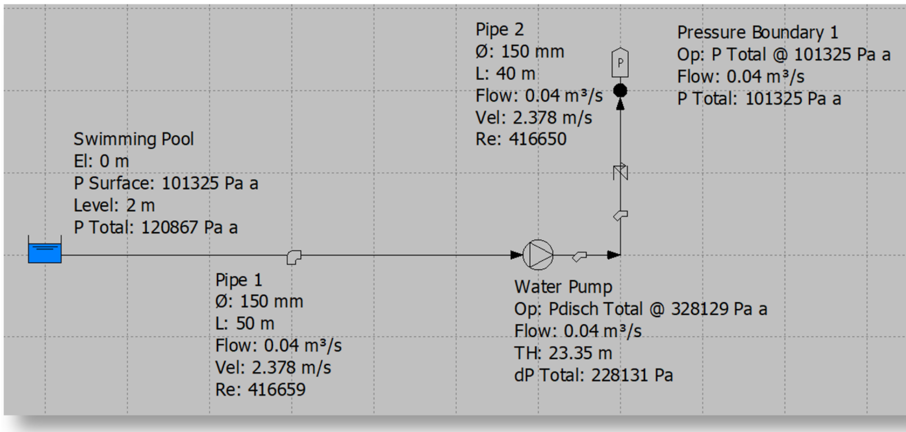


(A) Use a *Centrifugal Pump* operating with a constant *Discharge Pressure* to model the pump. Use a *Pressure Boundary* to model the pipe exit, maintaining a pressure of 101.325 kPa.

(B) What is the minimum *Discharge Pressure* of the pump required to maintain a flow rate of 40 L/s at the pipe exit? Build the system that was described, using the information provided in the figure above. Verify that you get the same results as seen in the figure below.

(C) Using Equation 7.4, manually verify that the $NPSH_A$ of the pump is approximately 9.847 m.

(D) Suppose that the pump is replaced due to wear and the new pump operates at a discharge pressure of 450 kPa. Propose a change that should be made to the system in order to maintain the desired flow rate of 40 L/s.



(E) The water park undergoes renovation and expansion and now the pump must feed two separate water slides at a height of 20 m. What changes must be made to ensure both water slides each maintain the desired flow rate of 40 L/s?

Problem #3. On page 80, it is mentioned that real-world (centrifugal) pumps tend to produce less of a pressure differential the greater the flow rate of fluid that passes through the pump. Let us model the water park's pump, now, with realistic dP vs. Q data.

(A) Record the total head produced by the pump and the volumetric flow rate that was set in the previous question.

(B) Remove the *Centrifugal Pump* device and replace it with a *Pressure Gain Device*. In the *Curve Data* dialog box, enter the following Q/dP data points:

Q (m^3/s)	dP (kPa)
0	206
0.032	163
0.063	148
0.095	131
0.126	112
0.158	81

Re-run the simulation. What is the total head produced by this pump with the realistic pump curve? How does this compare to the value you recorded in Part **(A)**?

(C) What is the actual flow rate of water produced by the pump? How does this compare to the value you recorded in Part **(A)**?

End of Tutorial

Tutorial #8:
Centrifugal Pumps

Tutorial #8 will consider the following concepts:

- Adding centrifugal pumps to the Flo-Sheet
- Installing pump catalogs
- Using and interpreting pump curves

In the previous tutorial, a *Pressure Gain Device* and a *Centrifugal Pump* were introduced. These devices are valuable tools for simulating *positive displacement* pumps and for modeling "generic" pumps (*i.e.* not much is known about the physical equipment). However, the *Pressure Gain Device* is not capable of considering factors such as the pump efficiency and the potential for cavitation. In this tutorial, we will explore the full potential of *centrifugal pumps* in fluid systems. In contrast to positive displacement pumps (see §7.2, page 85), many parameters must be considered in order to properly model centrifugal pumps. In particular, we will examine the relationship between key operational parameters including the pump head, $NPSH_R$, efficiency, and flow rate.

8.1 Centrifugal Pump Devices

Recall the water tower example from Tutorial #7: we used the *Pressure Gain Device* and *Centrifugal Pump* tools in order to determine the $NPSH_A$ and pump head (pressure change) required to supply the water tower.

At this point, reopen your Flo-Sheet (the pump and water tower) that you saved at the end of Tutorial #7. Save this as a new project file. Optionally, you can rebuild the Flo-Sheet from scratch for practice.

Suppose that we now know that the water tower's pump is centrifugal, by design, as seen in Figure 8.1. We will use Pipe-Flo to model this scenario. What is the pump's total head, $NPSH_A$, and the total volumetric flow rate?

Begin by deleting the *Centrifugal Pump* (or *Pressure Gain Device*) from your Flo-Sheet. In its place, add a new *Centrifugal Pump* from under the *Pumps* heading in the *Toolbox*. As seen in Figure 8.2, navigate to the *Property Grid* to specify the operational parameters of the pump. As in §7.2 we will specify parameters for the *Centrifugal Pump*, now including the *Pump Curve* field:

- The *Pump Curve*. The relationship between head/pressure differential/flow rate can be input manually or selected from Pipe-Flo's catalogs, if available.

For our water tower system, name the centrifugal pump and set the suction and discharge elevation to (both) 0 m: we will assume that there is no significant elevation change across the pump. This can be seen in Figure 8.3. Then:

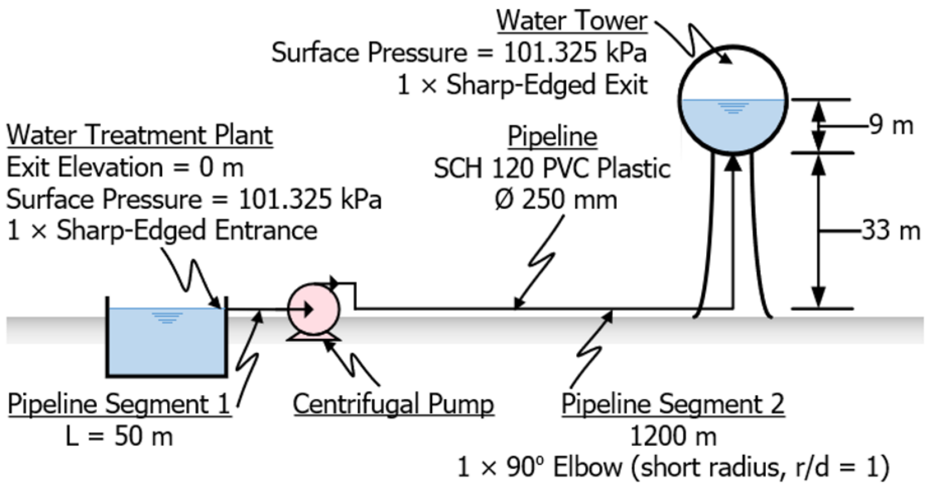


Figure 8.1: *The water tower from Tutorial #7 is now fed by a centrifugal pump.*

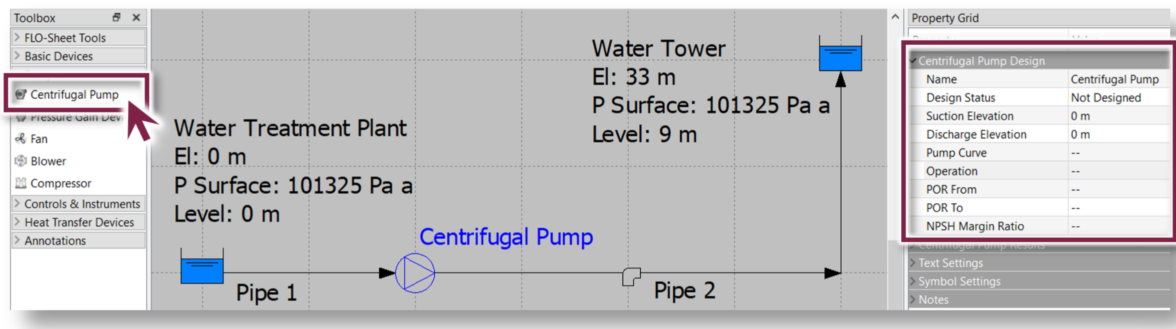


Figure 8.2: *The Centrifugal Pump tool is found in the Toolbox; edit its parameters in the Property Grid.*

- Click on the *Pump Curve* field ("Specify Pump"). A [...] icon will appear.
- Click on this icon; you will be given the choice to "*Select Catalog Pump...*" or "*Enter Pump Data...*"; then
- Choose the "*Select Catalog Pump...*" option.

As seen in Figure 8.4, the *Pump Selection* dialog box is then opened. It allows the user to choose a specific pump for their FLO-SHEET from one of PIPE-FLO's few pre-loaded pump catalogs. In each pump catalog, there are multiple different pump models with various operational characteristics such as rotational speed and flow rate. The user can download a vast selection of additional catalogs from the PIPE-FLO website at `https://cat.eng-software.com/user/sign-in/?return=%2Fproducts%2Fpump-flo%2Fpump-catalogs%2F`. Clicking the "Download Pump Catalogs" hyperlink in the *Pump Selection* dialog box will achieve the same effect.

⚠ The PIPE-FLO website will first ask the user to create a profile prior to downloading catalogs.

- Under the *Catalog* heading, the user can view pumps from the loaded catalogs and filter them by parameters such as "type" and "speed".
- Under the *Fluid Properties* heading, PIPE-FLO lists the properties of the fluid passing through the pump; these values are used, in turn, to calculate the pump's operational parameters.

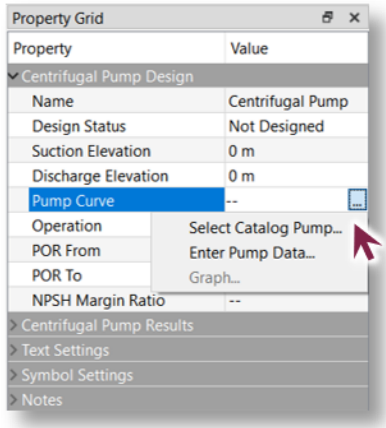


Figure 8.3: *Two options for entering curve data are available: selecting a pre-loaded pump or manually providing the data.*

- Under the *Search Criteria* heading, the user can filter the results from a particular (selected) catalog in order to find the ideal pump. If this feature is to be used, the "*Flow*" and "*Head*" must be specified. Optional search parameters include the "$NPSH_A$", "*Margin Ratio*", "*Preferred Operating Region*", "*Secondary Operating Point Flow*", "*Motor Enclosure*" and "*Motor Sizing Criteria*".
- Once a pump catalog is selected and the user has ticked at least one "*Type*" and "*Speed*" check box, all the available information on pumps that meet these criteria is tabulated in the lower half of the dialog box. This is clearly seen in Figure 8.4.

Now from the simulation results obtained in Tutorial #7 (see page 84) using a *Centrifugal Pump*, the total head required in order to maintain the flow rate of 0.1 m^3/s was 63.74 m. We will choose an actual centrifugal pump from PIPE-FLO's catalogs that meets these specifications.

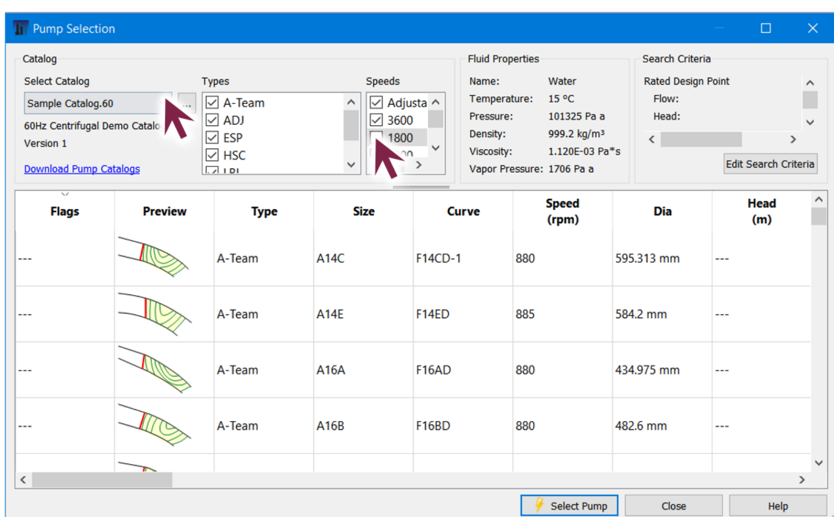


Figure 8.4: *The Pump Selection dialog box which will be blank upon opening for the first time. Here, pumps from PIPE-FLO's "60 Hz Centrifugal Demo Catalog" are displayed.*

- In the *Select Catalog* drop-down menu, choose the "60 Hz Centrifugal Demo Catalog" (*Sample Catalog.60*);
- Check all the pump *Types* and *Speeds*; then
- As seen in Figure 8.5, click Edit Search Criteria to specify the search criteria. Set the *Flow* to be 0.1 m^3/s and the *Head* to be 63.74 m. Click OK; we will not add any further parameters.

PIPE-FLO has refined the options in the *Pump Selection* dialog box to include only pumps that meet the stated criteria; in our example, there are 17 pumps that satisfy these constraints. We will choose the ESP (electric submersible pump) type pump of size 6×4–17 with a speed of 1,780 rpm. Double-click on the row belonging to this pump.

For these pumps, the size is given in the form $d_d \times d_s – d_i$ where d_d is the discharge port diameter, d_s is the suction port diameter, and d_i is the maximum nominal impeller size. These values are all given in inches.

Double-clicking on the pump opens a dialog box (as seen in Figure 8.6) containing the *pump curve* along with numerous important operational parameters. Find the location of the red knee (¬) in the pump curve. It highlights the *desired* operating point: 0.1 m^3/s and a head of 64 m, as seen under the *Data Point* heading. Clicking different points on the curve with your cursor (+) will display the performance characteristics (*e.g.* flow rate, $NPSH_R$, efficiency etc.) of the pump at that point.

The black line running through the knee is an imaginary *design curve* for the operating point calculated by PIPE-FLO to exactly match the required head and flow rate. This operating point/curve does not physically exist; it is merely the *minimum acceptable impeller diameter*. The *actual operating point* lies on the design curve of one of the available impellers from the manufacturer. That is, you want to choose the nearest operating curve (and that impeller size) *above* the thick black curve. To select the operating curve above the imaginary *design curve*, type the diameter from the curve into the *Curve* heading. Then, click Enter on your keyboard. This will move the black line and knee up to the specified impeller diameter, as seen in Figure 8.6. This is a crucial step and must be done in order to update your pump. Otherwise you will continue to operate with this imaginary pump and get improper results. Click OK.

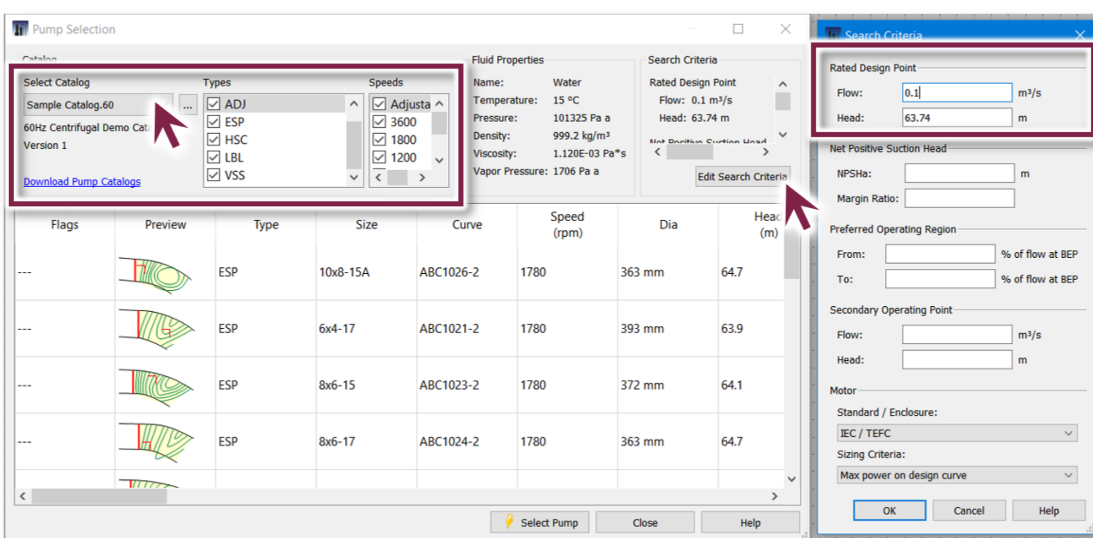


Figure 8.5: *Finding a centrifugal pump that is capable of moving 0.1 m^3/s of water while producing a head of over 63 m.*

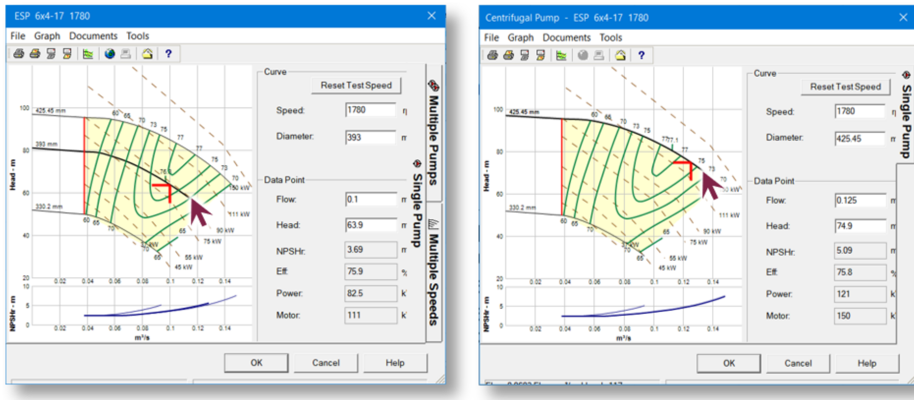


Figure 8.6: *Changing from the imaginary to the actually pump curve for the ESP 6×4–17 pump with a speed of 1,780 rpm and a diameter of 425.45 mm.*

Recall that the $NPSH_A$ from the previous tutorial was found to be 9.171 m (see page 84)—independent of the choice of pump. This particular centrifugal pump satisfies the important inequality $NPSH_A > NPSH_R$. As this pump is capable of (*i*) producing more than 63.74 m of head (*ii*) at a flow rate of 0.1 m^3/s (*iii*) while avoiding cavitation, it is an appropriate selection. Click on its row in the lower half of the dialog box then click Select Pump to finish. As a rule of thumb, given a number of acceptable options, choose the pump that *most closely matches* the operational criteria that you have specified in terms of head and flow rate, while considering the pump's efficiency.

Now that all of the design parameters have been specified for the centrifugal pump—including the pump curve—we are able to re-run the simulation. The results can be seen in Figure 8.7: the calculated flow rate and head values are 0.1253 m^3/s and 74.86 m, respectively. Also notice that the values of $NPSH_A$ and $NPSH_R$ are 8.654 m and 5.087 m; the simulation has determined that $NPSH_A > NPSH_R$, so cavitation will not occur. Or in other words, the suction pressure ($Psuct$ = 86,502 Pa) *exceeds* the vapor pressure of the water (P_{vp} = 1,706 Pa from the *Fluid Zones* dialog box). If you wish to go back and view the centrifugal pump's curve, there is an easy method of doing so. As seen in Figure 8.8, simply click on the pump, and then on the "*Graph*" icon on the ribbon (or type Ctrl+G). If *Calculation Mode* is enabled, the red knee (¬) will be visible, indicating the location of the operating point calculated by PIPE-FLO in the simulation.

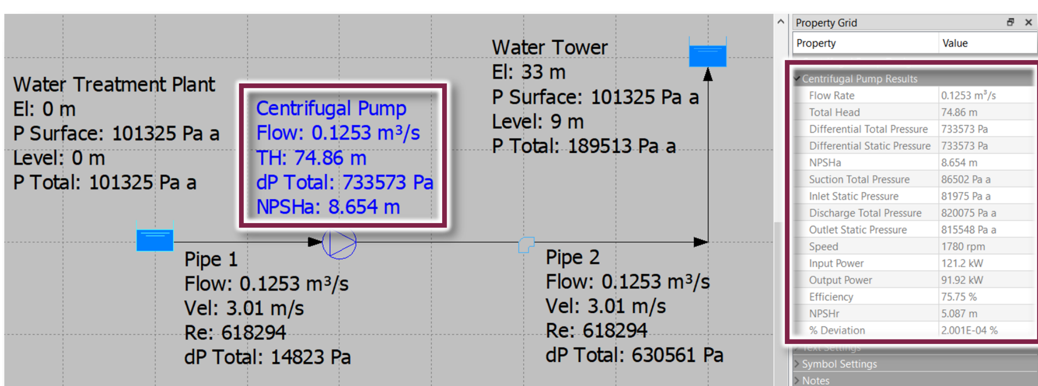


Figure 8.7: *Simulation results for the system with the 6×4–17 ESP pump operating at 1,780 rpm and the minimum acceptable impeller diameter.*

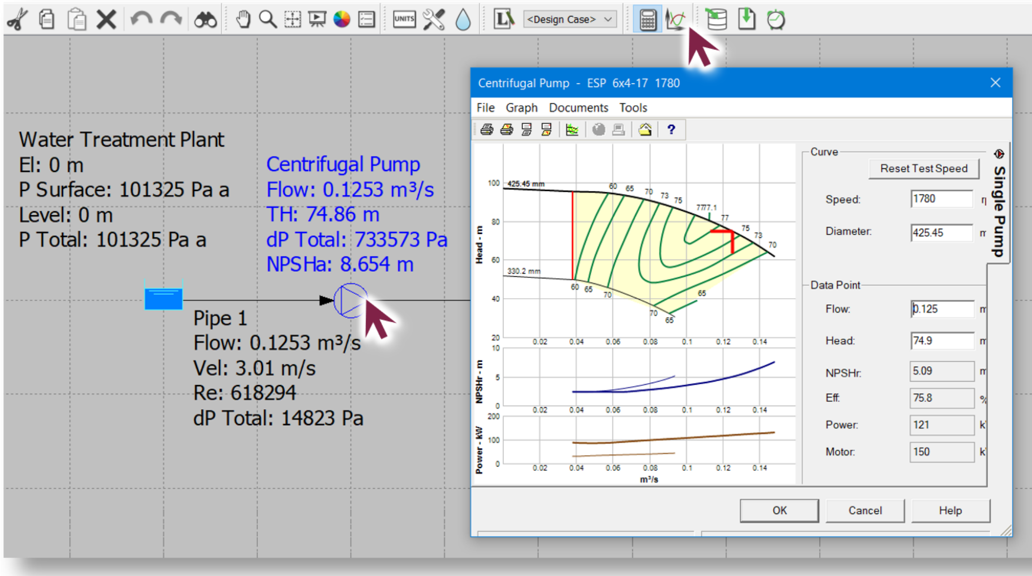


Figure 8.8: *With the pump selected, clicking on the "Graph" icon will display the pump curve.*

8.2 Pump Catalogs

Installing Pump Catalogs. As previously mentioned, PIPE-FLO maintains a large database of pump curves from over 100 manufacturers on its website. Use the URL on page 89 to access the database and add to the meager selection of pre-installed catalogs. In this tutorial, we will use *Kinetic Pumps*' catalog. For simplicity, it can be downloaded by accessing the following link: `latulippelab.mcmaster.ca/kinetic.exe`. To install the catalog, simply double-click on the file once it has downloaded. Click Close to complete the installation. The *Kinetic Pumps* catalog can now be accessed in the *Pump Selection* dialog box. If it does not appear right away, save your work, exit PIPE-FLO and reopen the software.

Interpreting and Using Pump Curves. In the simplest of terms, a pump curve is a visual representation of the relationship between the flow rate of fluid through a pump and its total head. These curves often summarize pump speed and efficiency data, as well.

In order to understand how to interpret and use pump curves, let us begin by creating a new project. On the blank FLO-SHEET, add a single *Centrifugal Pump* device.

As explained in §8.1, navigate to the *Pump Selection* dialog box. From the *Select Catalog* drop-down menu, select the "*Kinetic.60*" catalog: pumps manufactured by *Kinetic Pumps* which draw from a 60 Hz AC power source. Check all the boxes under the *Types* and *Speeds* headings in order to view all available *Kinetic Pumps* equipment. As seen in Figure 8.9, select the *Close Couple End Suction (EndSuct)* pump of size 2×1.25–5 operating at a speed of 1,750 rpm. Double-click to open this pump's curve.

Note that this pump curve is valid for water at 15.56°C, as per the "default" fluid specification under the *Fluid Properties* heading: no fluid zone has been specified! Also, no "knee" or "design curve" is shown because we have not specified the desired operating point!

As is visible in Figure 8.10, the upper graph is used to present the pump head versus flow rate through the pump:

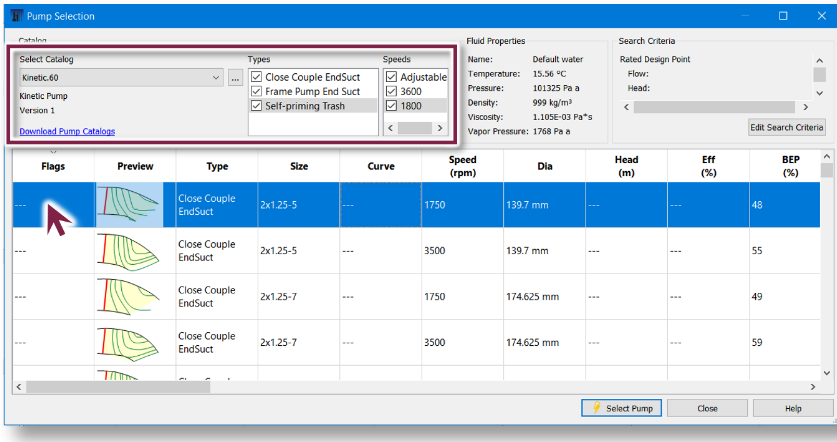


Figure 8.9: *The pumps contained in the Kinetic.60 pump catalog.*

- The top black curve represents the upper bound on head values produced by this pump with the 139.7 mm-diameter impeller. This is the largest impeller diameter for this pump offered by the manufacturer.
- Similarly, the bottom black curve represents the upper bound on head values produced by this pump with the 101.6 mm-diameter impeller. This is the smallest impeller diamter for this pump offered by the manufacturer.
- The thick green contour lines represent efficiency isolines. These are head/flow rate combinations at which the pump operates at the same efficiency η.
- The thick red line indicates the boundary of the minimum flow rate. Head/flow rate pairs to the left of the red line are unachievable with this pump. Contrarily, the yellow shading indicates the region in which the pump may operate effectively.

On the lower two graphs, the pump $NPSH_R$ (navy line) and pump power output (brown lines) are plotted versus volumetric flow rate. On the "power output" graph, the two lines correspond to the two pump impeller diameters plotted in the uppermost graph.

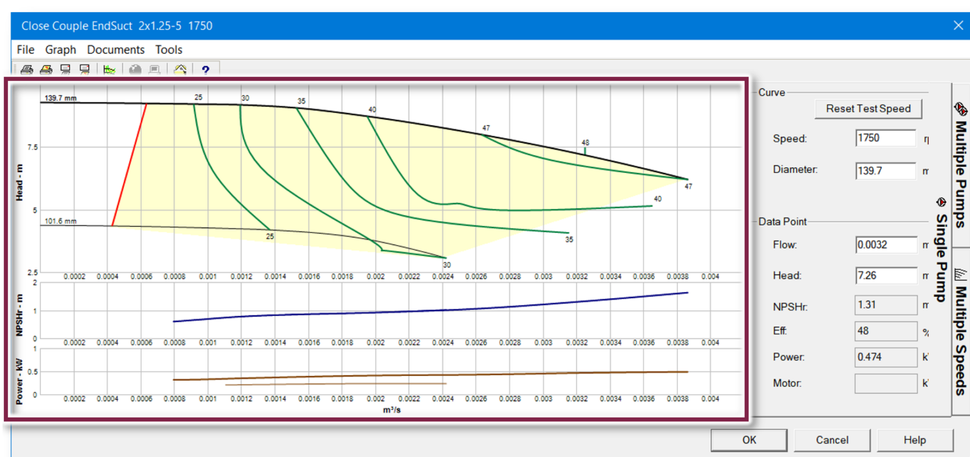


Figure 8.10: *Pump curve data for the "close couple end suction" pump of size 2×1.25–5 operating at a speed of 1750 rpm.*

ℹ Not all pumps in PIPE-FLO will have separate graphs for intrinsic relationships such as the power vs. flow rate curve. For example, Figure 8.8 displays the power vs. flow rate relationship as *isolines* directly on the pump curve.

Additionally, several other useful pieces of information are indicated on this graph:

1. **Shut-Off Head.** The shutoff head is the theoretical head produced by the pump when the flow rate is zero (*e.g.* a valve is closed). For the pump with the 139.7 mm-diameter impeller, this value is 9.3 m.

2. **Best Efficiency Point (BEP).** The BEP is the operating point where the greatest amount of head is produced per unit of energy added. This flow rate is the most efficient operating point. In the case of the pump with the 139.7 mm-diameter impeller, this corresponds to a flow rate of 0.0032 m^3/s, a pump head of 7.26 m, and an efficiency of 48%.

3. **Run-Out Point.** This is the maximum flow rate allowable, constrained by the design of the pump. Again in the case of the pump with the 139.7 mm-diameter impeller, the corresponding flow rate is approximately 0.0038 m^3/s.

ℹ If any of the above features are not displayed on your computer, open the *Graph Display* dialog box by clicking the [Graph Settings...] button or by navigating to [Graph » Display...]. As seen in Figure 8.11a, you will be able to toggle on/off such features including the efficiency curves, the $NPSH_R$ and power graphs, and lines representing other possible impeller diameters from the catalog. (Check/uncheck the *Catalog* box.) Click [OK] to finish.

As previously mentioned, if you are looking for more detailed information regarding a particular operating point, clicking on a graph with your cursor (+) will display the corresponding data values, as well. Instead of clicking on one of the graphs, you can directly input data values on the right side of the screen. For example, if you wanted to know the pump head if the flow rate was 0.0019 m^3/s (with the 139.7 mm-diameter and a speed of 1750 rpm), change the flow rate to 0.0019 m^3/s and click the [Enter] key on your keyboard. Figure 8.11b confirms that the pump head would be 8.79 m.

Finally, key pump data—such as operational limits—can be found in the *Design Notes* dialog box; navigate to [Documents » Design Notes] to open it. Under the *Pump Curve Data* tab, important information regarding this specific pump curve can be found, including the shut-off head, minimum flow rate, and best efficiency point. Under the *Pump Catalog Data* tab, general data about the pump model can be found, including the minimum/maximum speed, and temperature/pressure limits. This is illustrated in Figure 8.12b.

⚠ It is worth repeating again: if you have specified an operating point—the required flow rate and head—the red "knee" should appear on your pump curve along with the thick black "design curve". This design curve will be labelled with an impeller diameter, but beware that this curve does not necessarily correspond to an actual *available* impeller—this pump may not actually exist! The design curve represents the pump curve at the *minimum impeller diameter* needed to satisfy the operational requirements. So be sure that the *Catalog* option is selected to see the available impellers. The key takeaway here is that the design engineer must *select the next largest impeller* so to exceed the desired flow rate/head. Additional head losses can then be easily introduced by partially closing valves (*i.e increase* $\sum K$) in order to bring the system back down to the desired operational flow rate. Be sure to enter the next largest impeller diameter into the "*Diameter:*" field, and click the [Enter] button on your keyboard, as seen previously in Figure 8.6 and 8.11b. The system will recalculate with the new pump impeller.

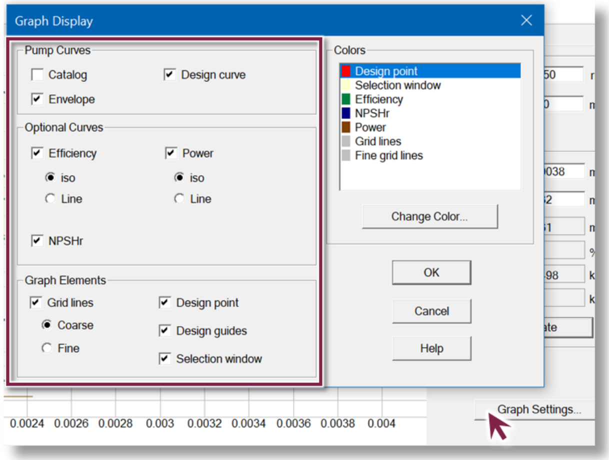


Figure 8.11a: *Changing the display settings for a given pump curve.*

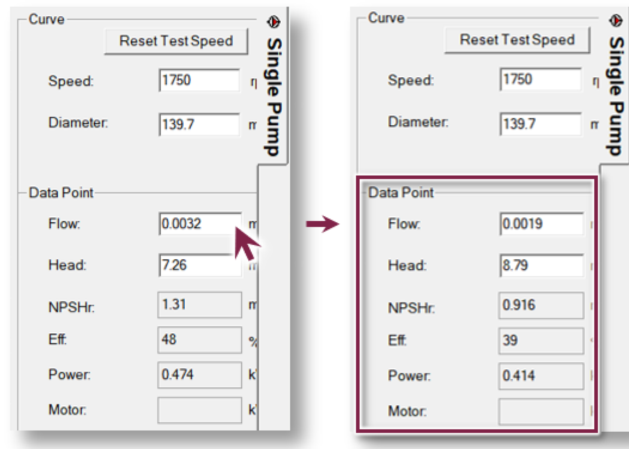


Figure 8.11b: Manually enter operating data in order to extract information from the pump curve.

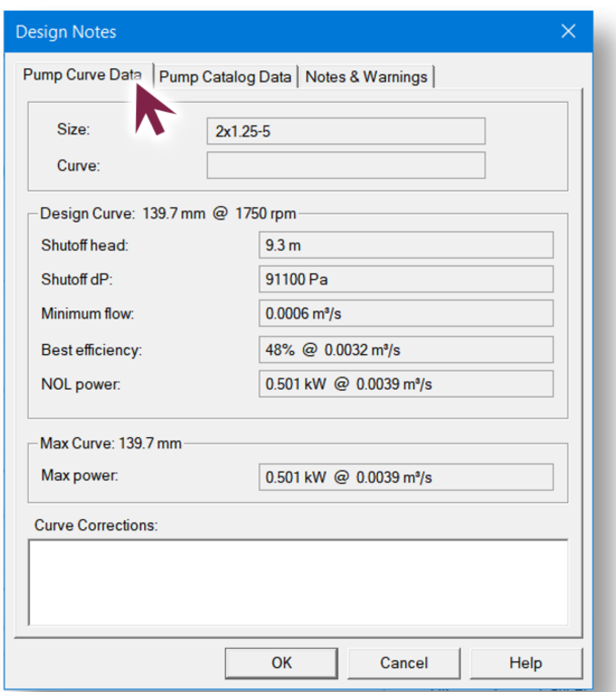


Figure 8.12a: *The Pump Curve Data tab includes information regarding the pump curve.*

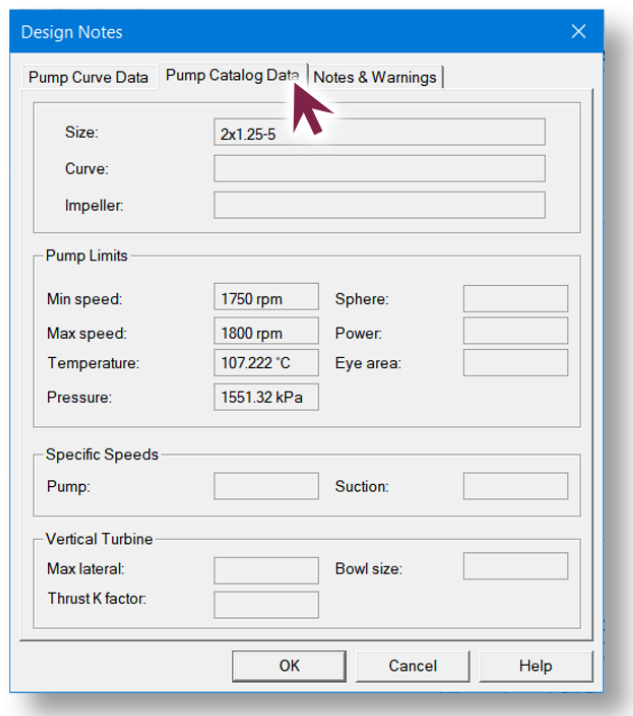


Figure 8.12b: The Pump Catalog Data tab includes general data regarding the pump

Tutorial Problems

The first two questions in these tutorial problems will all the use the following scenario:

As seen in Figure 8.13, an empty tanker truck is to be loaded with xylene (8°C)—a diesel fuel additive—from an underground storage tank. A centrifugal pump is used to pump the xylene from the underground tank and into the tanker through 65 mm diameter B241 aluminum pipes. Both feed and receiving tanks are open to the atmosphere (surface pressure = 101.325 kPa). In order to fill the tanker truck in a reasonable length of time, the flow rate of xylene into the truck must be 0.01 m^3/s. Note that the xylene is pumped into the *head space* above the liquid in the tanker truck. As the design engineer for the xylene delivery system, you must choose an appropriate centrifugal pump for this task. You may assume that there is no elevation change across the pump.

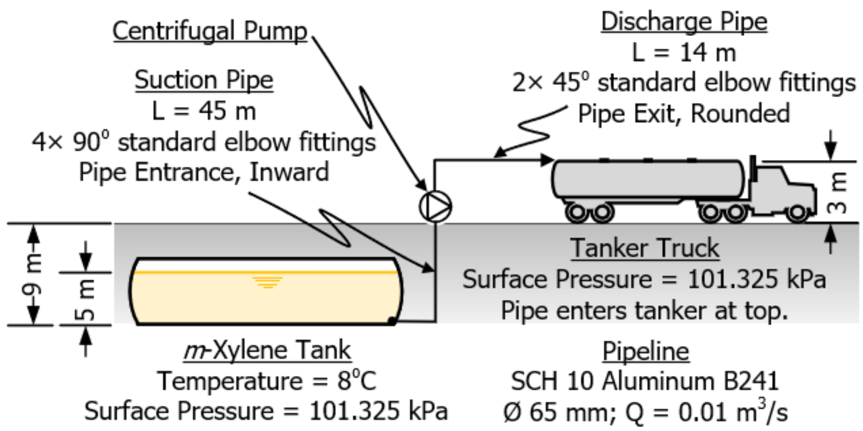


Figure 8.13: *A centrifugal pump is used to transfer xylene from an underground storage tank into a tanker truck.*

Problem #1.

(A) Build the system described in Figure 8.13. Consider the following:

- Choose "*m*-xylene" in the *Fluid Zones* dialog box to model this process.
- Choose a reasonable point of "zero elevation" and make sure you use it consistently when defining the elevation of other devices.
- Consider using a *Pressure Boundary* device to model the tanker truck instead of a *Flow Demand* device or *Tank*. *Why would this be recommended?*
- Add a *Centrifugal Pump* with a constant flow rate of 0.01 m^3/s. Since we are given a value for Q, no pump curve is required, and we will operate this pump like a *positive displacement pump* as described in §7.2, page 85.

(B) What is the pump's total head, $NPSH_A$ and differential pressure?

(C) If a xylene flow rate of 0.02 m^3/s is desired, is this feasible?

(D) From Equation 7.4, if the underground tank was pressurized (*i.e.* with gaseous xylene), theoretically, the desired flow rate in Part **(C)** could be achieved. Approximately, what surface pressure in the xylene tank would be needed to achieve a proposed xylene volumetric flow rate of 0.02 m^3/s?

Problem #2. Now, we will replace the *Centrifugal Pump* from the previous problem with a catalog pump. Reset your FLO-SHEET back to the original scenario from Part **(A)** of **Problem #1**.

(A) Using the *Centrifugal Pump*, load the *Kinetic.60* catalog in the *Pump Selection* dialog box. Allow PIPE-FLO to consider all types and speeds. Provide a ballpark estimate of how many 60 Hz *Kinetic* pumps that are available to you.

(B) In the *Search Criteria* dialog box, enter the following data:

- The required flow rate of 0.01 m^3/s;

- The total head that you found in Part **(B)** of **Problem #1**; and
- The $NPSH_A$ that you found in Part **(B)** of **Problem #1**.

To how many pumps has Pipe-Flo narrowed down the selection?

(C) Find the model of pump with the highest efficiency. Which pump is this, and what is the actual efficiency value? If there is more than one model with similarly high efficiencies, choose the "Frame Pump End Suct" model.

⚠ *Note:* Pipe-Flo will still display pumps with $NPSH_A < NPSH_R$. Be careful to choose a pump that will not cavitate!

(D) Select the pump from Part **(C)** and re-run your simulation. Are the actual simulated values of pump head and flow rate acceptable?

(E) Suppose that the desired flow rate of xylene is increased to 0.011 m^3/s. Using the pump curve, how does the pump head, $NPSH_R$, and efficiency? Is it possible to operate at this flow rate?

Problem #3. Using the *Sample.60* pump catalog, locate the VSS-type pump of size 3×1.5–10 with an impeller diameter of 254 mm operating at 1750 rpm. Answer the following questions about this pump.

(A) Locate the point of highest efficiency. What is this value (%), and to what flow rate/pump head does this value correspond?

(B) If the desired operating point is 0.005 m^3/s with a pump head of 25 m, is it feasible to operate this pump?

(C) If the impeller is reduced to a diameter of 152.4 mm, what is the point of highest efficiency? What is this value (%), and to what flow rate/pump head does this value correspond?

(D) Again, if the desired operating point is 0.005 m^3/s with a pump head of 25 m, is it feasible to operate the pump with this smaller impeller diameter?

(E) In the *Graph Display* dialog box, enable the "Catalog" option. What is the minimum impeller diameter needed in order to operate at the operating flow rate of 0.005 m^3/s and pump head of 25 m?

End of Tutorial

Tutorial #9:
Piping Networks

Tutorial #9 will consider the following concepts:

- Modeling piping systems with branches/tees and other complex elements
- Equalizing flows in parallel piping systems
- Integrating concepts from Tutorials #1–8 in piping networks

Calculating parameters related to flow through pipes by hand is a simple task for small systems. However, most real-world systems are exceedingly complex, making them impractical to model without a computer. As such, software tools like PIPE-FLO are implemented in order to determine factors such as pressure drops and pump sizes. These software tools automate the solving of "piping networks" by utilizing theories including:

- Kirchoff's Laws as they apply to piping systems
  - Law of Conservation of Mass
  - Law of Conservation of Momentum
- Pascal's Law
- The Hardy-Cross method, and other techniques applied to branched pipelines and elements connected in series/parallel

Now that we have learned about these individual elements that can be modeled by PIPE-FLO, let us combine some of them together in order create a more complex system: a piping network.

9.1 Modeling Piping Networks

As seen in Figure 9.1, a network of three heat exchangers (HX) is used to produce high-pressure steam which feeds a steam reformer in a process which generates gaseous hydrogen from natural gas. The HX network is fed at a flow rate of 4 m^3/s; the steam enters the steam reformer where the pressure at the top of the vessel (height = 8 m) of the vessel is 2,000 kPa. The pressure drop across each HX unit is known to be 72 kPa. It can be assumed that the steam temperature averages 500°C. We will neglect the effects of temperature changes. What is the required pressure of low-temperature steam being fed to the HX network?

Start by opening a new project FLO-SHEET.

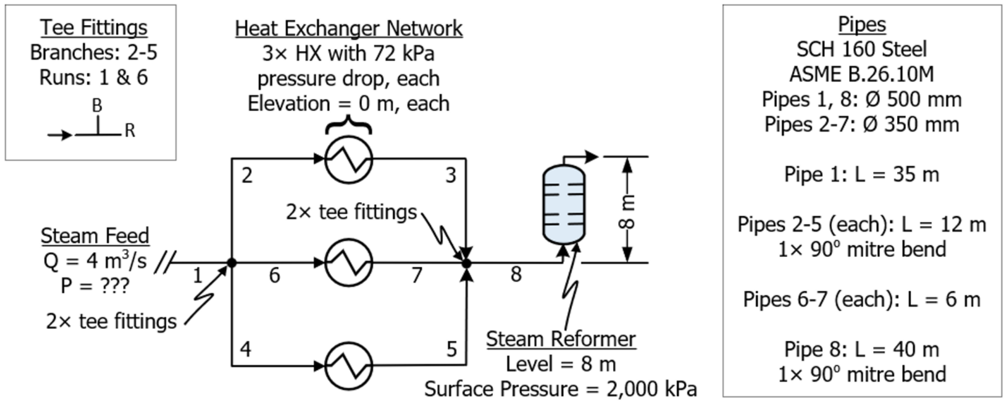


Figure 9.1: *A heat exchanger is used to produce high-pressure steam to feed a steam reformer.*

Step One: Set up the system.

- Add a "steam" fluid zone at 500°C and 2,000 kPa.
- Create pipe specification for SCH 160 Steel ASME B.36.10M pipe.

Step Two: Build the system.

- Start by using a *Flow Demand* device to model the feed entering the system.
- *Nodes* can be used to create a "split" in a pipeline. Connect a pipe to the *Flow Demand* and leave a node at the downstream end.
- Add the three HX units to the FLO-SHEET. They will be modeled as *Fixed dP Devices*. As we know what our desired flow rate is, we will assume that the pressure drop will also be fixed.
- Connect the open node to the *Fixed dP Devices* with pipes.
- In a similar manner, connect another pipe to one of the *Fixed dP Devices* and leave a node at the other end.
- Connect the two remaining *Fixed dP Devices* to this node with pipes.
- Add a *Tank* device to the FLO-SHEET to model the steam reformer.
- Connect the last node to the *Tank* with one final pipe.

Step Three: Specify the system parameters.

- Set the flow rate at 4 m^3/s for the *Flow Demand* device. Assume an elevation of 0 m, and set the *Flow Type* to *Flow In*.
- On each of the *Fixed dP Devices* (HX), set the fixed pressure drop to be 72 kPa. Assume that the inlet and outlet elevation of these HXs are all 0 m.
- For the *Tank* device (steam reformer), set the surface pressure to be 2,000 kPa and the liquid level to be 8 m. Also assume that the elevation of the reformer is also 0 m.
- Set the elevation of all the nodes to be 0 m.
- Ensure that the *Fluid Zone* for all components of the FLO-SHEET is the "steam" zone that you created earlier.
- Set all the pipes to be SCH 160 Steel ASME B.36.10M.

  - ▷ For Pipes 1 & 8, set their nominal diameters to be 500 mm; Pipes 2–7 should have a nominal diameter of 350 mm.
  - ▷ Pipe 1 should be 35 m long; Pipes 2–5 should be 12 m long; Pipes 6 & 7 should be 6 m long; finally, set Pipe 8 to be 40 m long.

- To account for minor losses: add one 90° mitre bend (each) to Pipes 2–5 and Pipe 8. Modeling the losses that occur at the pipe tees (where the pipes split into multiple directions):
  - ▷ In the *Valves and Fittings* dialog box, add *two* "*Tee - Flow Thru Run*" fittings each to Pipes 1 and 7. As seen in Figure 9.1, the *run* of a tee fitting is the *straight* path passing through the tee.
  - ▷ Similarly, add one "*Tee - Flow Thru Branch*" fitting (each) to Pipes 2–5. Also seen in Figure 9.1, the *branch* of a tee is the path where the fluid must turn (change direction) while passing through the fitting.
  - ▷ We will ignore the effects of the pipe contractions/expansions as we enter or leave the system.

> Every time there is a "branch", the flow through the "run" is affected due to turbulence. In this system, we require *four Tee - Flow Thru Run* fittings because physically, *four* tee fittings are needed: one for each branch (Pipes 2–5). As Pipes 1 and 7 branch *twice each*, we need two *Tee - Flow Thru Run* fittings in these segments.

Step Four: Changing the *Fixed dP Device* icon. Users in PIPE-FLO are actually able to change what the icons on the FLO-SHEET look like. For example, the *Fixed dP Device* uses the "" symbol. Typically, heat exchangers are represented by the "" symbol. To make this change, right-click on the *Fixed dP Device* icon and choose the *Change Symbol...* option from the menu. As seen in Figure 9.2, select the "" option. Click OK. *This procedure works with any icon on the* FLO-SHEET.

For reference, your FLO-SHEET up until this point should appear like the one illustrated in Figure 9.3. Now, we are ready to run the simulation!

> Before you run the simulation, make a hypothesis about what will happen. How will the volumetric flow rate and fluid velocity compare in each of the three HX "streams"?

To answer our original question, from Figure 9.4, we see that the feed pressure of steam to the bank of HX units is 2,085 kPa.

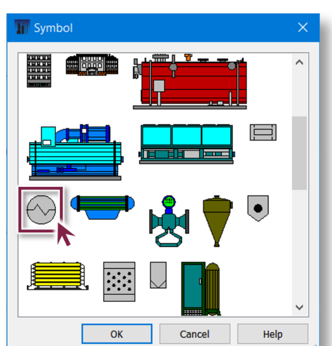


Figure 9.2: *Changing the icon representing the heat exchangers on the* FLO-SHEET.

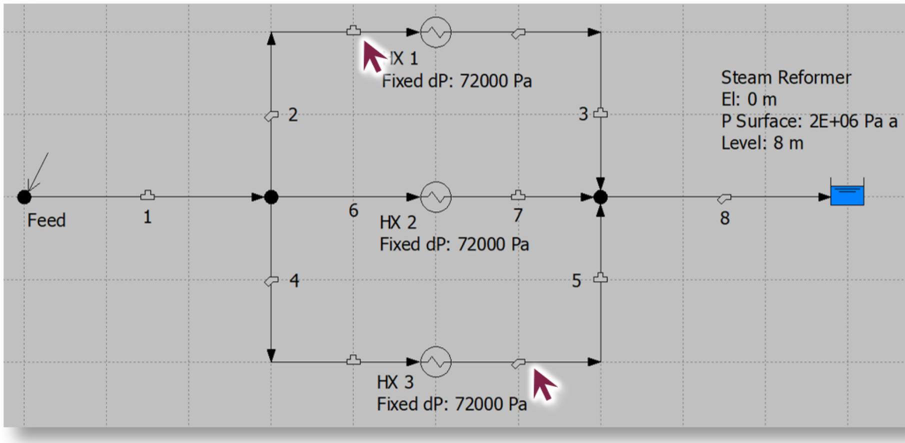


Figure 9.3: *The* FLO-SHEET *modeling the HX network, where each HX has a pressure loss of 72 kPa. Note the "tee" and "bend" symbols displayed on the pipes.*

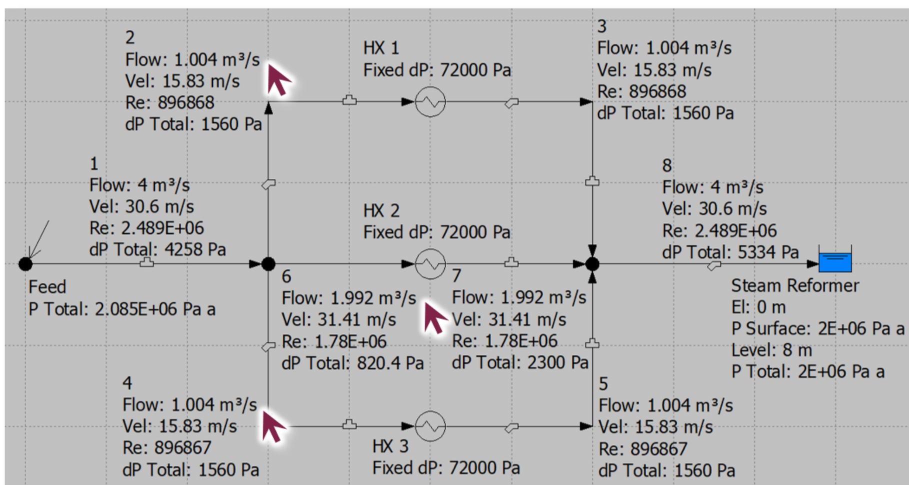


Figure 9.4: *The results from the simulation involving the HX network.*

9.2 Balancing Piping Networks

Balancing Flow. In Figure 9.4, what do you notice about the flow rates of steam passing through HX units 1, 2, and 3? The outer HX units (1 & 3) each receive essentially one half of the amount of steam that the central unit (2) receives. On its face, this is odd because all of the HX units are identical. However on closer inspection, the pipes connecting HX1 and HX3 to the network incur substantially greater minor losses than the pipes connecting HX2 to the network. Specifically:

- **HX1 & HX3.** Pipes 2–5 each experience $\sum K$ values of 1.577.
- **HX2.** Pipes 6 and 7 have $\sum K$ values of 0 and 0.5256, respectively.

From this, it should be clear to you that the pipes attached to HX1 and HX3 experience a greater *pressure drop* (due to the pipe bends and flow through branches) than the pipes attached to HX2. As a result, there is a *greater resistance* to fluid flow in the outer HX units, and thus a greater fraction of the steam passes through the central HX unit where the resistances are lower. *Like electricity, fluids will flow through the path of least resistance.* In this scenario, though, that is not a desirable outcome. Due to the increased flow rate, HX2 will need to produce a greater *heat duty* to raise the temperature of the steam, while HX1 and HX3 will require small heat duties. This mismatch in heat duties is inefficient from design and operational standpoints. One strategy that we can use to balance this system is to add a *Control Valve* device to the pipe feeding the central HX unit:

- Place a *Control Valve* on Pipe 6, splitting it in to two pieces of equal length.
- Specify the elevation of the *Control Valve* as 0 m.
- In the *Operation* dialog box, set the *Flow Rate* to a fixed volumetric flow rate of 4 $m^3/s \div 3 = 1.333$ m^3/s.

You are now ready to re-run your FLO-SHEET. The results should be identical to those shown in Figure 9.5. You can see that (within rounding error) the flow rates entering the three HX units are now equal. Much better!

Adding a Turbine. Let us add an addition level of complexity to our system. As seen in Figure 9.6, suppose that high-pressure steam that was generated with our bank of HX units needs to be distributed to *two* different processes, now: the steam reformer unit and a steam turbine. The turbine is used to recover additional energy from the system that might otherwise be wasted. The turbine is fed from the same node as Pipe 8, the pipe which feeds the steam reformer. Additionally, the turbine requires an inlet pressure of 1,500 kPa. Let us add onto our previous simulation to determine the effect of this change.

- Add a *Pressure Boundary* device to the FLO-SHEET to model the turbine. Set its *Elevation* and its *Pressure* to be 0 m and 1,500 kPa, respectively.
- Connect the *Pressure Boundary* device to the node at the nexus of Pipes 3, 5, 7, and 8. Use the same pipe specification as before, but with a diameter of 350 mm and a length of 100 m.
- To the pipe, add minor losses associated with the 15- 90° mitre bends and the *Tee - Flow Thru Branch* fitting.
- Add *one more Tee - Flow Thru Run* fitting to Pipe 7 as the inclusion of the turbine adds another branch onto this pipe.

Now, re-run the simulation; the result is shown in Figure 9.7. Note the *Level 3* errors associated with the reversed flow through Pipe 8 leading to the steam reformer, and through pipe 9 leading to the Turbine. Why is this occurring? Recall from the previous system, fluids will take the path of least resistance. This is what is occurring with the first error. The pressure requirement for the turbine is much lower than the pressure in the rest of the system. As such, the high-pressure steam held in the reformer seeks to flow to the lower-pressure turbine. To prevent this reversed flow, we can use another *Control Valve* to control the pressure in the system:

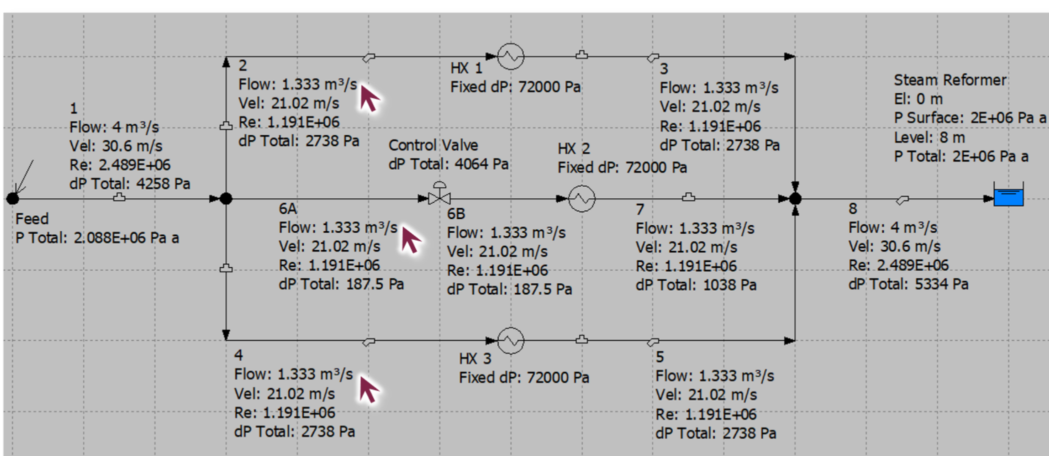


Figure 9.5: *The results from the simulation involving the HX network with included Control Valve devices.*

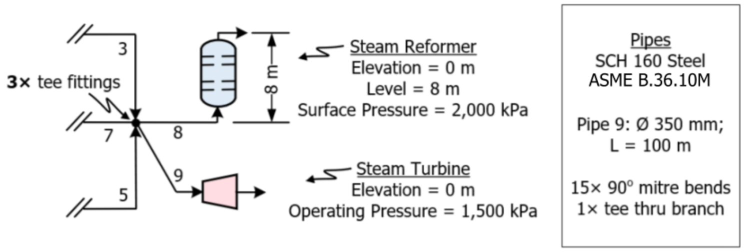


Figure 9.6: *Both the reformer and a turbine require the high-pressure steam generated by the bank of heat exchangers.*

- Perform another *pipe split* operation on Pipe 9—the pipe leading to the turbine.
- Here, we will arbitrarily split Pipe 9 such that the first 99 m of the pipe is upstream of the new node, while the last 1 m is downstream of the node. Assign all minor losses to the upstream portion of the pipe.
- Add a *Control Valve* device to the new node.
- Set the valve's *Elevation* to be 0 m, and assign the *Discharge Pressure* to be 1,500 kPa.

At this point, we can re-run the simulation once more. The results are shown in Figure 9.8. As you can see, there is still an error regarding the pressure drop ratio in Pipe 9B. Why is this occuring? Since steam is a compressible fluid, and there is a significant pressure drop across *Control Valve 2*, the steam properties change significantly. As a result, the properties of the *Fluid Zone* that we defined at the beginning of the tutorial poorly represent that actual properties of the steam exiting the *Control Valve*. Recall that the density and viscosity of gases change significantly with pressure and temperature. To fix this error, we will create a low pressure fluid zone to place in Pipe 9B:

- Add another "steam" *Fluid Zone* at 500°C and 1,500 kPa.
- Change the *Fluid Zone* in Pipe 9B to be this low pressure steam.

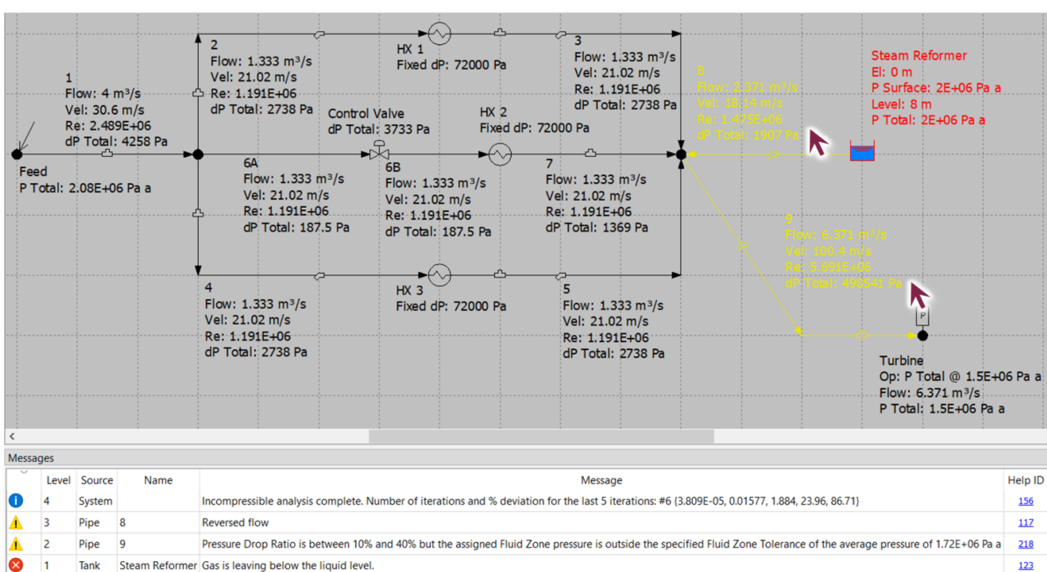


Figure 9.7: *Results for the system with a reformer and a turbine, with an indication of reversed flow in the pipe leading to the reformer.*

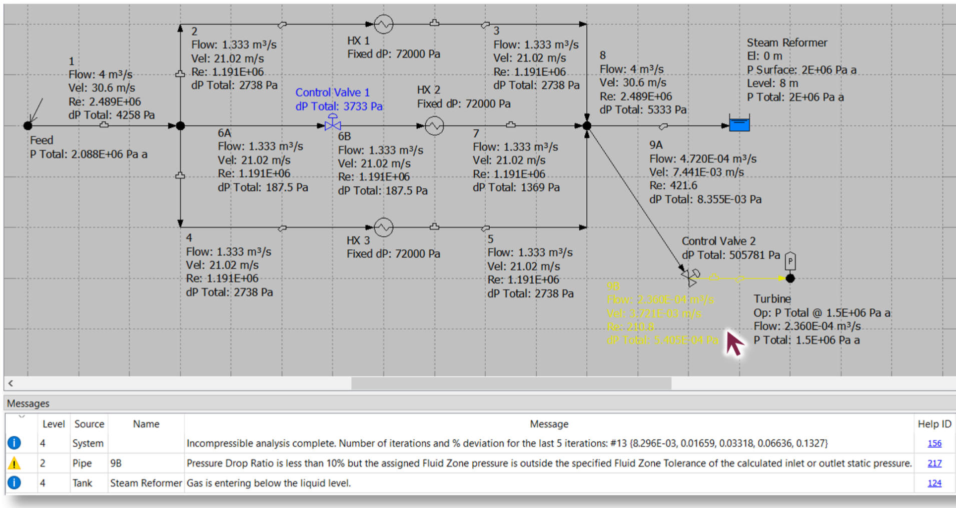


Figure 9.8: *Results for the system with a reformer and a turbine, balanced using a back pressure valve.*

As visible in Figure 9.9, no errors are incurred in the simulation! The *Control Valve* constricts the flow through Pipe 9A/B, maintaining a pressure of 1,500 kPa on the downstream side of the valve.

And there we have it: a balanced piping network with a bank of HX units feeding a reformer and a turbine. PIPE-FLO enables the design and simulation of complex piping systems.

Be sure to save your PIPE-FLO simulation. We will be using the FLO-SHEET in the tutorial questions.

Additional Considerations. The design and analysis of piping systems is a complex, multifaceted task. An in-depth knowledge of balancing orifices, control valves, and other process equipment is crucial to perform such tasks. Most of these intricacies are well beyond the scope of this course, but the key takeaway is that software tools such as PIPE-FLO and ASPEN PLUS can be used to efficiency simulate extremely complex piping systems that would be infeasible to model by-hand.

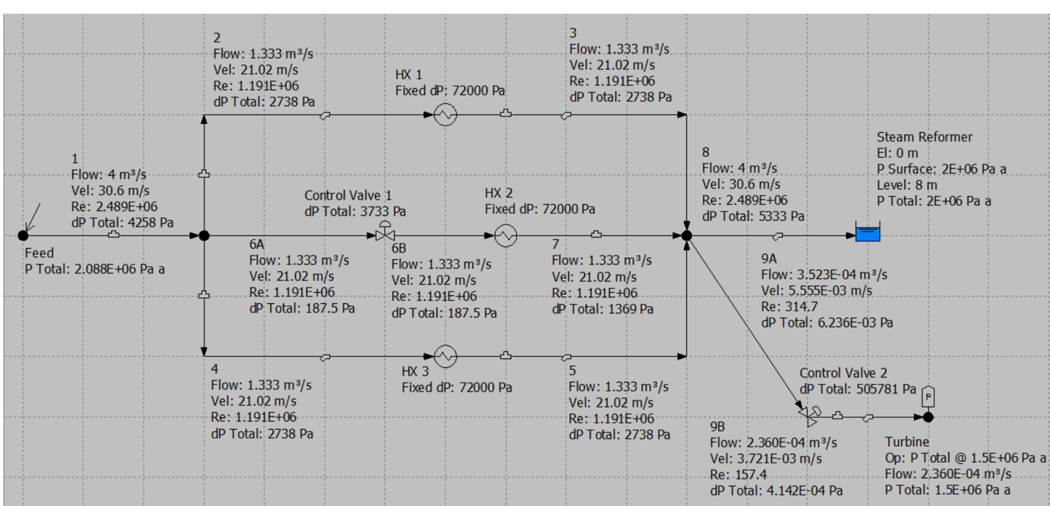


Figure 9.9: *Final results for the system with HX units, a reformer and a turbine.*

Tutorial Problems

Problem #1. Using the FLO-SHEET that was completed earlier in the tutorial (see Figure 9.9), what are the effects of the following changes on the system? Note: changes from previous parts of the question *do not* carry through to subsequent parts!

(A) Suppose that the uppermost HX unit, HX1, is taken offline for maintenance. What happens when Pipe 2 is closed to access HX1? To completely block or close a pipe:

- Right-click on that pipe and select "*Close Pipe*"; or
- Select the *Close Pipe* tool from the *Toolbox* then click on the pipe that you wish to close.

(B) Suppose that HX2 is taken offline for maintenance. What happens when Pipe 6A is closed?

(C) Suppose that production in the steam reformer is halted. What happens when Pipe 8 is closed to prevent the reformer from being fed with steam?

(D) Suppose that HX3 becomes fouled from particulates in the feed, causing it to create a fixed pressure drop of 80,000 kPa. What happens to the system?

(E) Suppose that HX3 becomes heavily fouled, causing it to created a fixed pressure drop of 110,000 kPa. What happens to the system?

(F) There is a flow surge in the system and the feed flow rate increases to 6 m^3/s. How does this affect the system?

Problem #2. A closed-loop refrigeration system uses six heat exchanger (HX) units to chill six refrigerated chambers. The refrigerant, 1,2-ethanediol, is pumped from a storage tank through a complex piping network and back into the storage tank. The system and its design parameters are presented in Figure 9.10. The key design details are as follows:

- The process fluid is 1,2-ethanediol at -7°C and 200 kPa.
- All the lines are constructed of STD-schedule B302 copper pipe. The nominal pipe size is $\varnothing$ = 50 mm for all the lines.
- The minor losses inherent to each pipe have been tabulated as a single $\sum K$ value (*V&F K*). Input each of these *V&F K* values in the *Valves and Fittings* dialog box as a "*Fixed K*" under the *Other* heading. No other minor losses are exhibited!
- The HX units are modeled as *Fixed dP* devices, each with a pressure drop of 15 kPa.
- The *Centrifugal Pump* device operates at a *Discharge Pressure* of 450 kPa.
- All devices on the FLO-SHEET have an *Elevation* of 0 m.

Build this FLO-SHEET as described, and answer the following questions. Notice that the minor losses associated with the three HX units (HX4–HX6) on the right side of the FLO-SHEET have *double* the $\sum K$ values compared to the three HX units (HX1–HX3) on the left side of the FLO-SHEET. *Hint*: You can build the first three HX units then copy/paste them to create the last three HX units.

(A) What is the total flow rate of refrigerant moved by the pump and the total pump head?

(B) What is the fraction of the total flow that is sent to the first three HX units (HX1–HX3)? What is fraction of the total flow that is sent to the last three HX units (HX4–HX6)?

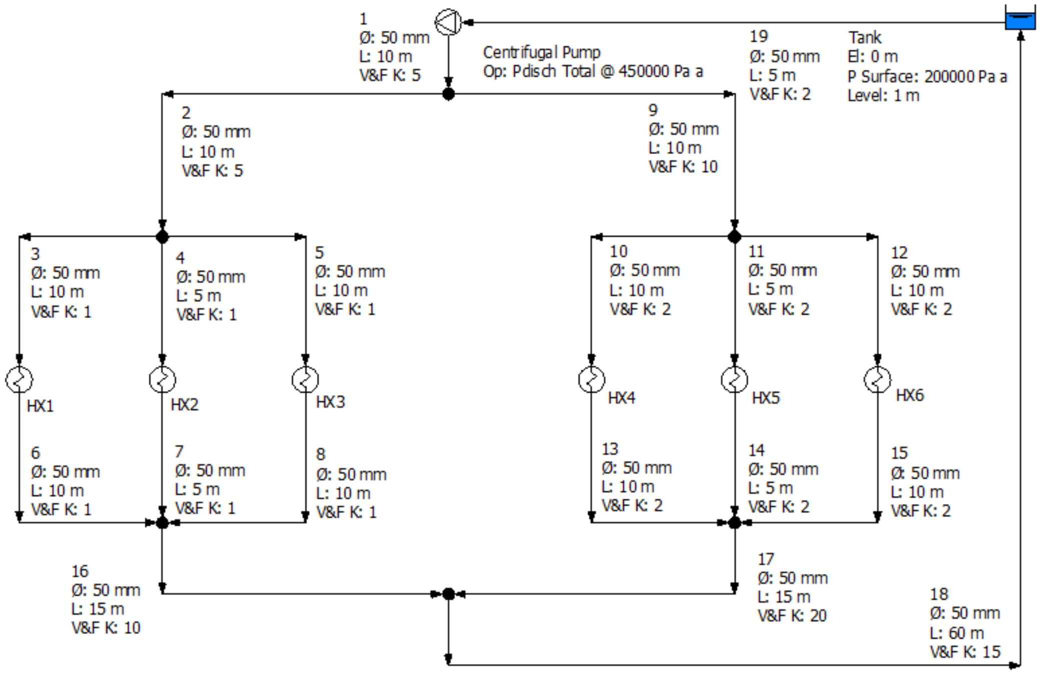


Figure 9.10: *A closed-loop refrigeration system is used to chill six chambers. Note: this is an image of the* PIPE-FLO FLO-SHEET *taken in Presentation Mode. See page 7 for details.*

(C) Suppose that the pressure drop across units HX1–HX3 is *permanently* increased to 25 kPa? How does this change your answers in Parts **(A–B)**?

(D) Keeping the change made in Part **(C)**, replace the *Centrifugal Pump* device with *any* fixed-speed centrifugal pump that meets the demands of the system. Record the pump's key characteristics such as its name, size, speed, efficiency, and $NPSH_R$.

(E) Re-run the system with your new centrifugal pump. Record the flow rate and pressure change across the pump as well as the total pump head.

(F) Balance the flow rate of refrigerant entering the last three HX units (HX4–HX6) with the flow rate of refrigerant entering the first three HX units (HX1–HX3) by adding a *Control Valve* device to the FLO-SHEET. That is, ensure that the *total* flow rate of refrigerant entering HX1–HX3 (combined) matches the *total* flow rate of refrigerant entering HX4–HX6 (combined). With a centrifugal pump, you will need to *iterate several times* by changing the valve's flow rate set point and observing the response! Note any changes to the flow rate and pressure change across the pump as well as the total pump head.

End of Tutorial

Tutorial #10: The *Saturn V* Rocket

Tutorial #10 will consider the following concepts:

- Integrating concepts from Tutorials #1–9 in one comprehensive example
- Using the PIPE-FLO software in complex real-world systems to calculate system design parameters

Pictured in Figure 10.1, the *Saturn V* rocket was operated by NASA from 1967–1973 in the *Apollo* lunar exploration program. Notably, it carried the *Apollo* astronauts to the moon. At the time of writing, the *Saturn V* rocket is the largest, heaviest, and most powerful rocket ever to have been operated. In this tutorial, we will use PIPE-FLO to perform a fluid transport simulation on the propellant feed system on the F-1 rocket engine located in the first stage of the *Saturn V* rocket. In these simulations, we will utilize real data made available by NASA to generate a single *realistic* system to model. All the provided values and specifications are assumed to be directly obtained from NASA and subcontractor design specifications; estimated/approximated values are marked with a superscript dagger†. In particular, we will use PIPE-FLO to answer the following question: "how much power was required to operate the fuel/oxidizer turbopump found in each of the *Saturn V*'s five main-stage rocket engines?"

Figure 10.1: *The Saturn V rocket. Image courtesy of NASA.*[1]

This final tutorial is designed to reinforce the concepts taught in previous tutorials. As such, each individual concept will not be covered at the usual level of detail. As one final note, this tutorial does not feature the "usual" tutorial problems found at the end of Tutorials #1–9. It is recommended that you view the following as the "culminating exercise" of this courseware.

[1] https://www.nasa.gov/sites/default/files/thumbnails/image/ksc-69pc-442.jpg

10.1 Background

> Do not become too mired in the details, here. The modeling process will be clearly outlined in the subsequent section, and almost everything that you need is summarized in Figure 10.6.

As illustrated in Figure 10.2, the *Saturn V* rocket was designed as a three-stage expendable rocket with the ultimate goal of transporting man to the moon. Each of the three stages was designed for a specific purpose and contained its own rocket engine or sets of engines:

- **S-IC:** *S-IC* was the first (bottom-most) stage of the *Saturn V* rocket. This stage stood at 138 feet long and 33 feet in diameter, with a total weight of 4,792,200 pounds at launch. *S-IC* primarily consisted of kerosene (RP-1) fuel and liquid oxygen (LOX) oxidizer tanks which fed the five F-1 rocket engines which were needed to propel the rocket to an altitude of approximately 38 miles above the Earth's surface at a speed of 5,250 mph.
- **S-II:** *S-II* was the middle stage of the *Saturn V* rocket. At 81.5 feet long and 33 feet in diameter, this stage weighed 1,140,000 pounds at launch. This stage was designed to continue to propel the rocket upwards after *S-IC* consumed all its fuel and was subsequently jettisoned. Like *S-IC*, *S-II* consisted mainly of the liquid hydrogen (LH_2) fuel and LOX oxidizer tanks which fed the five J-2 rocket engines needed to propel the rocket to an altitude of 109 miles and a speed of 15,647 mph.
- **S-IVB:** *S-IVB* was the upper stage of the *Saturn V* rocket. At 58 feet 7 inches long and 21 feet 8 inches in diameter, this stage was the smallest, weighing in at 262,000 pounds at launch. After *S-II*'s fuel was exhausted and the stage was jettisoned, *S-IVB* boosted the payload (service module, command module, and lunar module) into orbit about Earth in preparation for a lunar intercept. In addition to the three crewed modules, this stage carried liquid hydrogen (LH_2) fuel and LOX oxidizer tanks for use by its single J-2 rocket engine.

Here, we will focus on the lower stage of the rocket, *S-IC*, seen in Figure 10.3. As is visible, *S-IC* housed a 768 inch-tall LOX tank and a 517 inch-tall RP-1 tank separated by a 262.4 inch "intertank section". According to NASA's *SA-503 Saturn V Flight Manual*:

- The fuel tank was loaded with 203,000 gal of RP-1 at ambient temperature (80°F†) with a surface pressure of 24.2 psia.
- Similarly, the LOX tank carried 331,000 gal of liquid oxygen maintained at -297°F with a surface pressure of approximately 18–20 psia.

The 2219-aluminum fuel and oxidizer tanks (center diameter = 33 ft) sat atop the 233.7 inch-tall thrust structure which housed the five Rocketdyne F-1 engines. Illustrated in Figure 10.4, each F-1 engine drew the bipropellant mixture of RP-1 and LOX via the turbopump from the storage tanks.

- RP-1 was transported to *each* engine from the bottom of the fuel tank through two 12-inch suction lines constructed of 6061-aluminum†.
- LOX was drawn by each engine from the bottom of the oxidizer tank through one 17-inch 6061-aluminum† suction line.
- Each suction line contained a simple butterfly† "pre-valve" used to cut off the flow of fuel/oxidizer in case of an emergency.

> Given the lack of additional information, we will assume that all duct diameters are given as *nominal diameters*†. We will also assume that this value is close to the pipe inner diameter due to the large size of these lines.

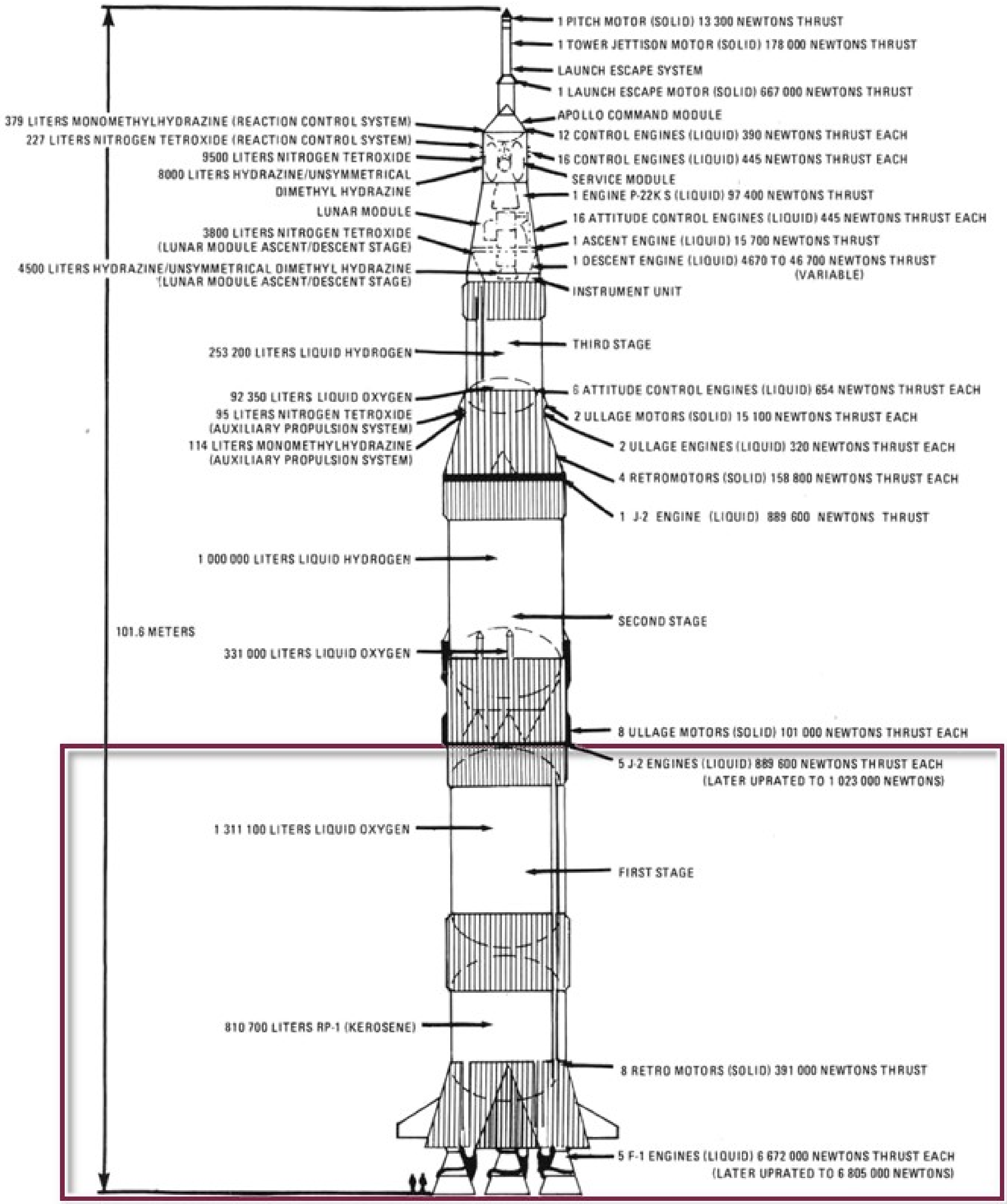


Figure 10.2: *A schematic of the Saturn V rocket. Stage S-IC is indicated. Image courtesy of NASA.*[2]

[2] https://history.nasa.gov/SP-4206/p405.jpg

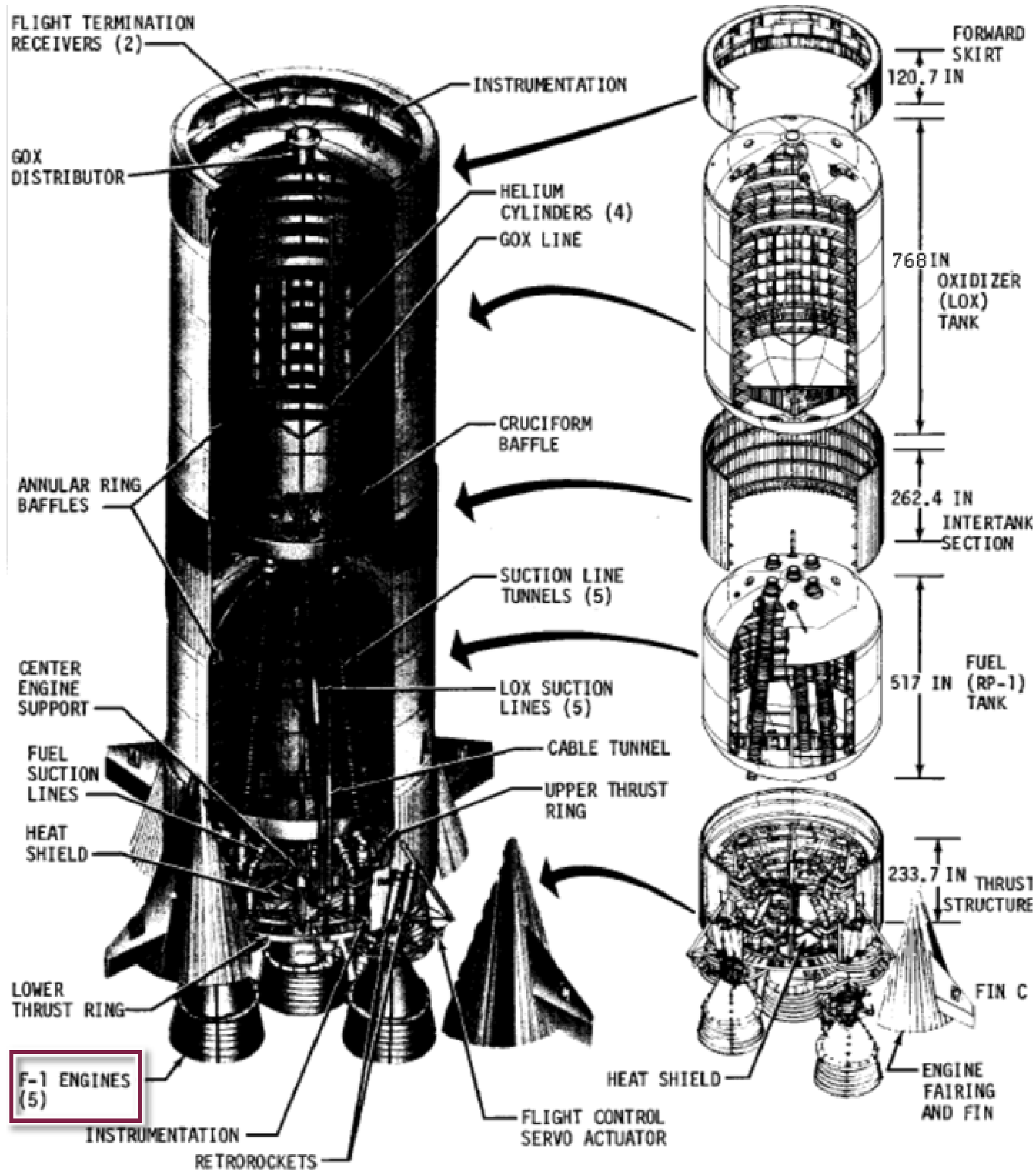


Figure 10.3: *A schematic of S-IC with the F-1 engine highlighted. Image courtesy of NASA.*[3]

Each engine was fitted with its own turbopump used to draw the fuel and oxidizer from their storage tanks, pressurize it, and feed it to the engine's thrust chamber to be combusted. In reality, the engine's turbopump actually consisted of *two* separate fuel/oxidizer pumping chambers (*volutes*) connected to a gas turbine by a common shaft. In the rare *F-1 Rocket Engine Technical Manual*, Rocketdyne and NASA provide the following normal-operation parameters with regards to *each of the F-1 engines in S-IC*:

- The turbopump impeller located in the fuel volute had a diameter of 22.85 inches.
- The turbopump impeller located in the oxidizer volute was slightly smaller, at 19.5 inches.
- As both impellers were mounted on the same shaft, they rotated at the same speed which was nominally 5,500 rpm.

[3] *Saturn V Flight Manual SA 503.*

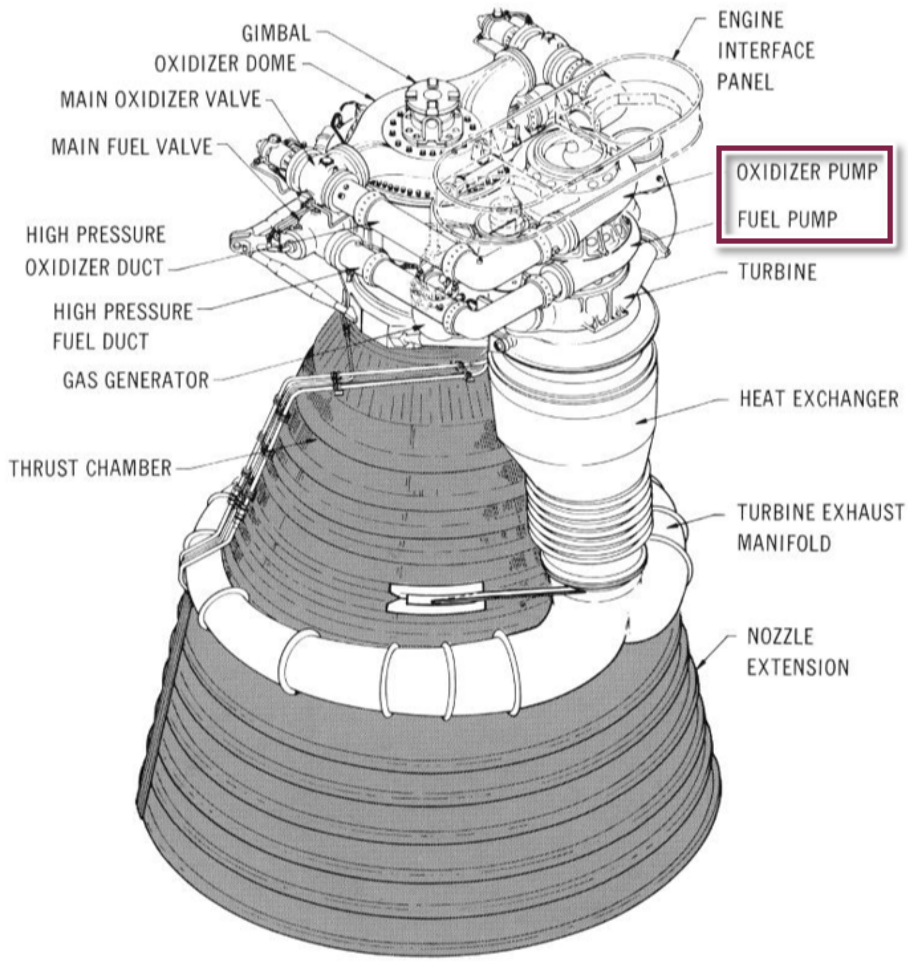


Figure 10.4: *A schematic of the Rocketdyne F-1 engine. The turbopump components are highlighted. The image originates from NASA.*[4]

After being expelled from the turbopump volutes, the fuel and oxidizer were fed to the thrust chamber through duplicate (*i.e.* two for the fuel and two for the oxidizer) high-pressure ducts:

- Both the high-pressure fuel and oxidizer discharge ducts (two each per engine) can be assumed to have had a 6-inch† diameter and were made of 6061-aluminum piping†.
- The flow rates of fuel and oxidizer to be injected into the thrust chamber were controlled using the "L-shaped" main fuel and oxidizer control valves:
  - In order for the F-1 engines to provide adequate thrust, it was required that the flow rate of fuel to each engine was 15,632 gallons per minute (gpm).
  - Similarly, it was required that the flow rate of oxidizer (LOX) to each F-1 engine was 25,038 gpm.
- In the thrust chamber, it was known that the pressure and temperature were 1,125 psi and 5,970°F.
- The fixed pressure loss associated with the fuel injector was 96 psi.
- Similarly, the fixed pressure loss associated with the oxidizer injector was 312 psi.

[4] http://heroicrelics.org/info/f-1/f-1-with-callouts.html

Given these system design parameters and operational constraints, we would like to know the steady-state *power required* by the turbopump in each of the F-1 engines. How many horsepower will the gas turbine have to expend in order to run the turbopump at the required pressures/flow rates?

10.2 Modeling & Simulation

We will now build a Pipe-Flo simulation in order to calculate the horsepower rating of the turbopump in the F-1 engine; create a new project and Flo-Sheet. Before we get started, you may have noticed already that the design parameters for each F-1 engine have all been listed in Imperial units. As such, it is a good idea to change the default units in your project to match those given in this tutorial. The recommended units are shown in Figure 10.5. Once the default units have been set, we can begin the modeling phase.

Step One: Specify the fluids used in this system.
RP-1 Fuel. Create a *Custom Fluid* type for RP-1 as it is not listed in any of the Pipe-Flo databases. As an important note, RP-1 is a mixture of hydrocarbons and it is difficult to assign individual values for properties such as its vapor pressure. Thus, we will have to make estimations/assumptions about some of RP-1's properties.

- RP-1 is a liquid in this system. From measurements of a surrogate kerosene mixture, its critical pressure can be estimated as 320 psi†.
- Approximate the fluid temperature to be the temperature of RP-1 in the fuel tank: 80°F†.
- Approximate the fluid pressure to be the surface pressure of RP-1 in the fuel tank: 24.2 psi.
- According to a NIST study, the density of RP-1 (at 80°F and 24.2 psi) is 49.7 lb/ft^3.
- From the same NIST data source, the absolute viscosity of RP-1 is 1.446 centipoise (cP).
- Using a kerosene analogue, it is estimated that the vapor pressure of RP-1 is 0.20 psi†under these conditions.

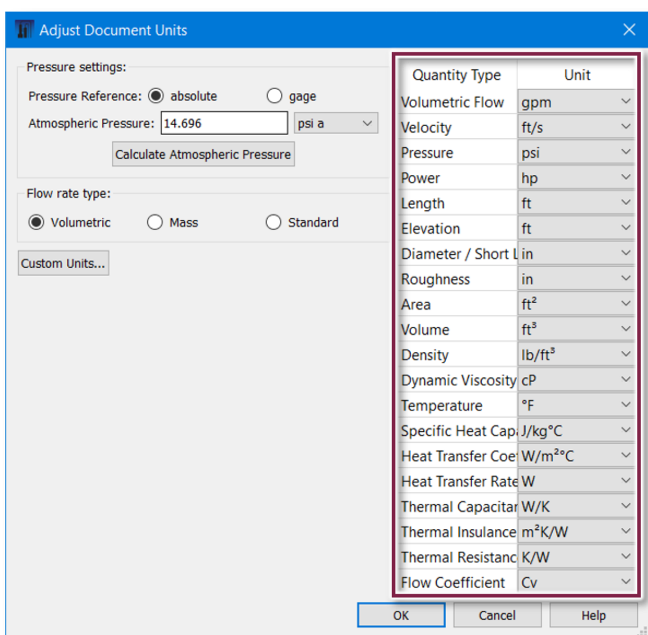


Figure 10.5: *It is recommended that you change your values to be displayed in these (Imperial) units.*

Liquid Oxygen Oxidizer. Luckily, LOX is much easier to model in Pipe-Flo as it is already pre-loaded in the *NIST* data table. Specify an oxygen fluid zone, as follows:

- Set the temperature of the oxygen to be -297°F, the temperature of LOX in the oxidizer tank.
- Set the pressure of this fluid zone to be 19 psi, the average surface pressure of LOX in the oxidizer tank.

Step Two: Create the pipe specifications. Like working with RP-1 fuel, the pipes used on the *Saturn V* rocket were non-standard. Therefore, we have to be a little creative with our piping specifications. All of the suction and high-pressure discharge were likely constructed of a 6061-aluminum† alloy (and/or 2219-aluminum† alloy). By default, Pipe-Flo does not include these types of aluminum pipes—especially at large diameters—in the piping specifications catalog. Luckily, Pipe-Flo *does* service ASTM B345 ANSI H35.2 aluminum. The takeaway here is that both 6061-aluminum and ASTM B345 ANSI H35.2 aluminum have the *same roughness value:* 6×10^{-5} in. While the actual inner pipe diameters of ASTM B345 ANSI H35.2 aluminum will not be exactly the same as the ducting on the rocket, it will be close enough for our purposes. Thus, create a specification for *SCH 40 ASTM B345 ANSI H35.2 aluminum piping.*

Step Three: Assemble the model. Use the information presented in §10.1 as well as the schematic in Figure 10.6 to design your Flo-Sheet. The following list contains a series of key assumptions that will need to be made, as well as other design considerations:

- The tank liquid levels were estimated using tank geometries and fluid properties. These are "initial" values: the liquid levels *before liftoff.* Intuitively, the volume of fluid in the fuel/oxidizer tanks will decrease with time.
- For the 17-inch LOX suction line (Pipe 4), use the *18-inch nominal diameter pipe* as a 17-inch pipe does not exist in the pipe tables. The 18-inch pipe is closest to the desired specifications. Otherwise, assume that the diameters given in the courseware correspond to the "nominal" pipe diameters available in Pipe-Flo.
- The pipes of "negligible" length (Pipes 3, 5, 10–12, 17 & 18) act as "placeholders" in the Flo-Sheet. These pipes did not truly exist in the real *Saturn V* rocket, but are merely used here to connect different elements together. As such, we do not want them to influence the solution. Choose a small arbitrary length for these pipes: 0.1 ft will suffice.
- The "butterfly pre-valves" are modeled as minor losses (*i.e.* valves and fittings) in Pipes 1, 2 and 4.
- Use nodes when two pipes join to become one, or where one pipe splits to become two. On the entire Flo-Sheet, ignore minor losses due to junctions (tees; branches and runs).
- Use *Centrifugal Pump* devices to model the turbopump volutes. *For now, do not assign the design parameters.*
- For simplicity, there is no elevation change across the turbopump and from the turbopump to the thrust chamber. All components have the same elevation from the turbopump, onward.
- The main valves (fuel and oxidizer) can be modeled as a control valve in order to control the flow rate of fuel and oxidizer leaving the pump.
  - Each high-pressure fuel line should maintain a steady-state flow rate of 15,632 ÷ 2 gpm.
  - Similarly, each high-pressure oxidizer line should maintain a steady-state flow rate of 25,038 ÷ 2 gpm.
- The injector losses can be modeled as *Fixed dP* devices.
  - Losses associated with each high-pressure fuel line: 96 psi.
  - Losses associated with each high-pressure oxidizer line: 312 psi.
- Use *Pressure Boundary* devices (4) to model the thrust chamber pressure at the end of each high-pressure discharge line.

At this point, you should have enough information to specify almost everything in the system: the tanks, pipes, nodes, control valves, pressure-drop devices, and pressure boundaries. The only underspecified pieces of equipment on your Flo-Sheet should be the two centrifugal pumps (one for each volute).

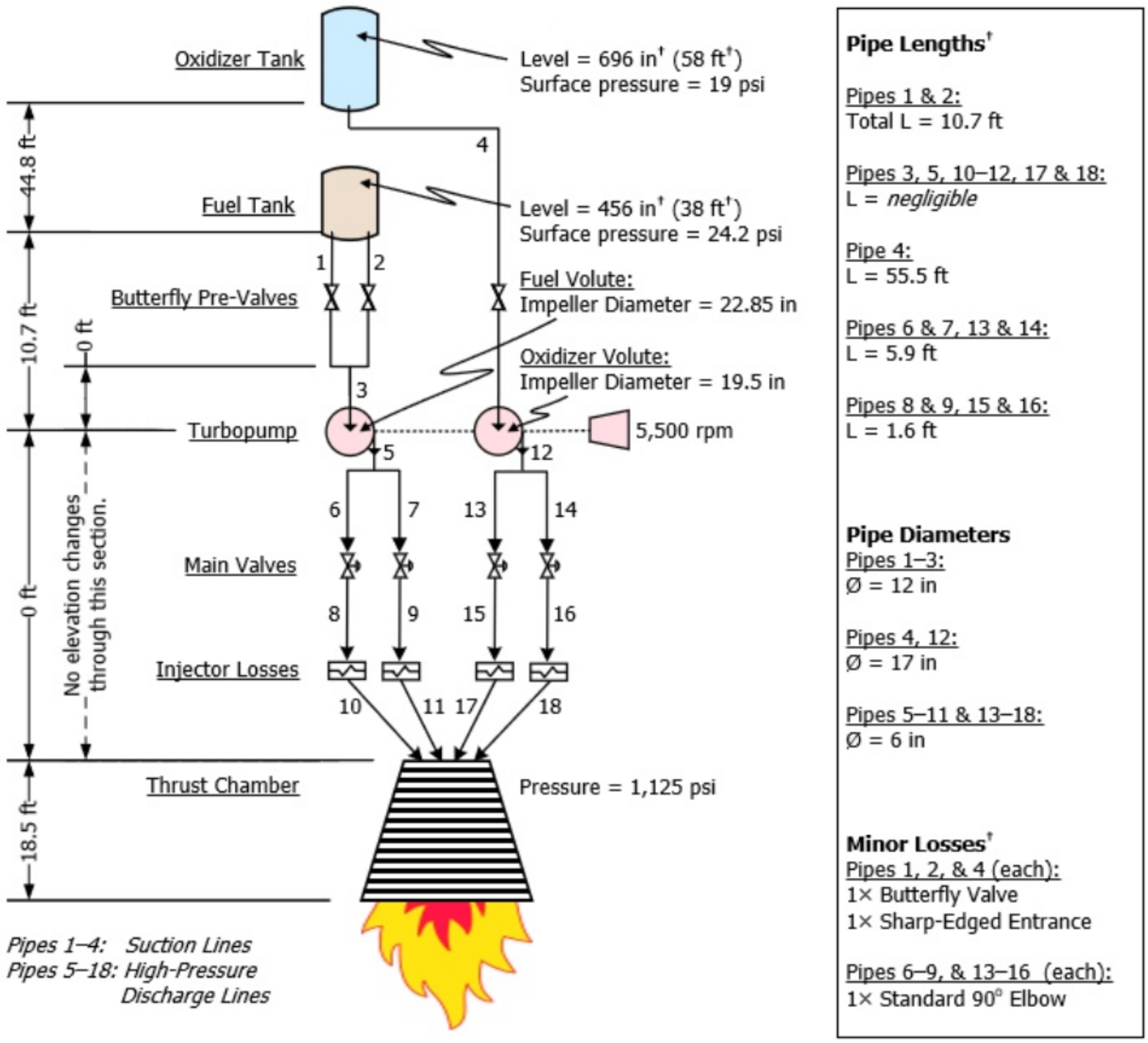


Figure 10.6: *Schematic of the Saturn V S-IC propulsion system to be modeled using* Pipe-Flo.

Step Four: Specify the turbopump. In order to accurately model the turbopump, we require its pump curve. Luckily, NASA published the pump curves for both turbopump volutes in its *F-1 Rocket Engine Technical Manual*, as seen in Figure 10.7. As both curves are essentially linear, we can model the curves as linear relationships over the operational range of the turbopump. Therefore we only require to pass Pipe-Flo two (three, counting a zero flow rate) additional flow rate/pump head/efficiency triplets per centrifugal pump. These points—extracted from the curves—are summarized in Table 10.1. Also included in the table are approximate volute efficiency values for the turbopump, obtained via actual F-1 turbopump efficiency measurements and theoretical correlations utilized by NASA.

Using these values, we can now fully specify the turbopump volutes. Individually select the *Centrifugal Pump* devices and do the following:

- Ensure that the *Suction Elevation* and *Discharge Elevation* are equal and at the same elevation as all the components downstream of the turbopump volutes;
- Click on the *Pump Curve* field ("Specify Pump");
- Click on the ... icon and select "*Enter Pump Data...*";

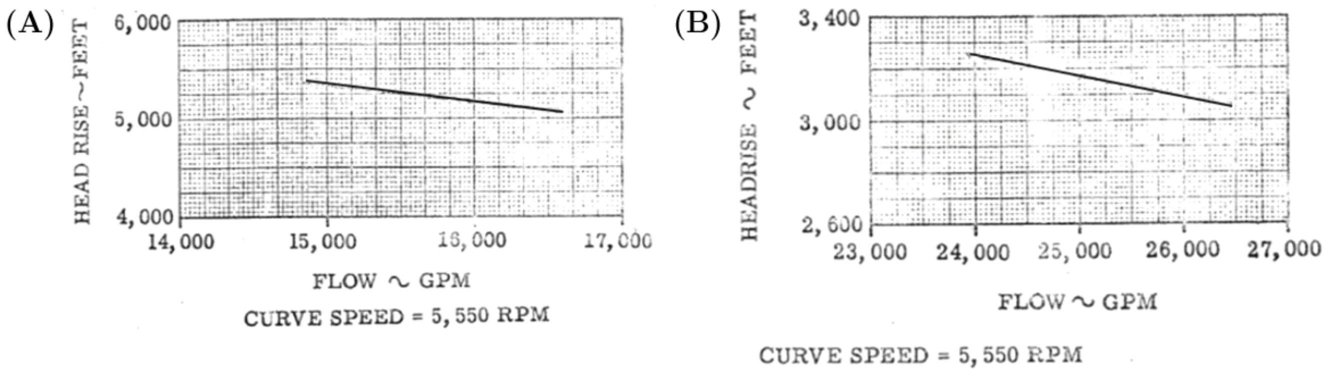


Figure 10.7: *Actual pump curves (head vs. flow rate) for the Rocketdyne F-1 turbopump.*[5] *Both the fuel* ***(A)*** *and oxidizer* ***(B)*** *volute curves are shown.*

- In the *Enter Pump Data* dialog box, fill in:
  - The pump *Test Speed* (5,500 rpm);
  - The impeller *Diameter* (22.85 or 19.5 in); and
  - The flow rate, pump head, and efficiency data points corresponding to the correct turbopump volute which are summarized in Table 10.1.
- Set the pump *Operation* to be "*Fixed Speed*";
- Click OK to finish. Be sure to specify both turbopump volutes.

Your completed FLO-SHEET should appear similar to that shown in Figure 10.8.

Step Five: Run the simulation. We are now ready to simulate the system! Enter into *Calculation Mode* and ensure that no errors have been indicated. If you discover that there are errors when you simulate, go back and review **Steps 1–4** to ensure that you are using the correct values and parameters for the components on the FLO-SHEET. Notice the following key results:

- Fuel is fed to the pump (through Pipes 1 & 2) at a combined flow rate of 15,632 gpm—exactly as specified. Similarly downstream of the turbopump fuel volute, the combined flow rate in the pipes is exactly 15,632 gpm. The two main fuel valves each generate a pressure drop (dP) of 577.1 psi each in order to obtain this flow rate of RP-1.
- A similar finding is obtained on the oxidizer side of the FLO-SHEET. LOX is fed to and from the turbopump at a flow rate of 25,038 gpm. The two main oxidizer valves each generate a pressure drop (dP) of 43.88 psi in order to obtain this flow rate of LOX.
- The discharge pressure of the fuel and oxidizer turbopump volutes are 1841 and 1632 psi, respectively. This is very close to actual values reported by NASA: 1,857 and 1,598 psi for the fuel and oxidizer volutes!

Table 10.1: *Flow rate, pump head, and efficiency data for the F-1 turbopump fuel and oxidizer volutes.*

	Fuel Volute			**Oxidizer Volute**	
Flow Rate (gpm RP-1)	Pump Head (ft)	Efficiency† (%)	Flow Rate (gpm LOX)	Pump Head (ft)	Efficiency† (%)
0	8,350†	0	0	5,050†	0
15,000	5,350	71.8	24,000	3,250	73.2
16,000	5150	73.1	26,000	3,100	75.8

[5] *F-1 Rocket Engine Technical Manual R-3896-1.*

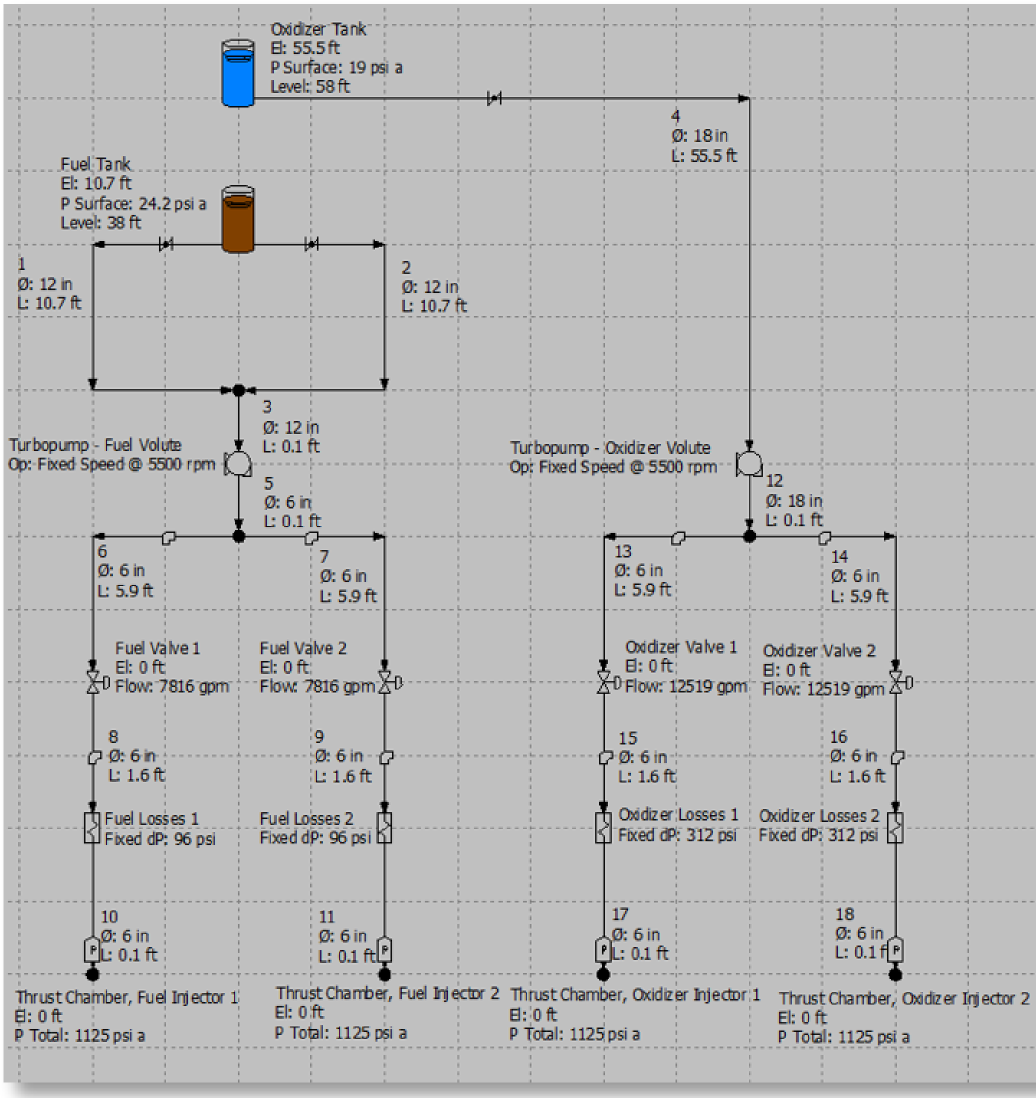
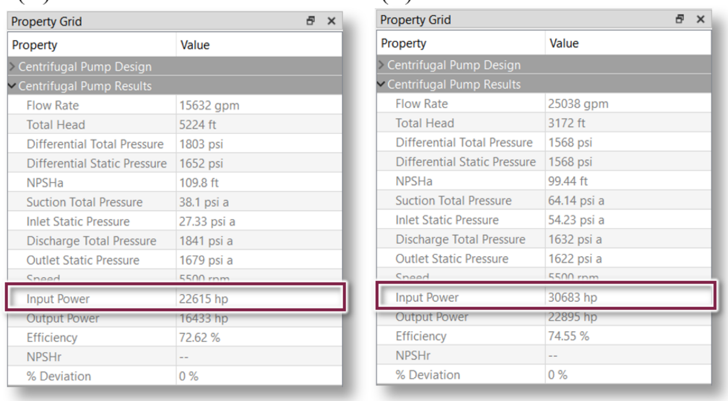


Figure 10.8: *Completed* FLO-SHEET *modeling the S-IC propellant feed system on the Saturn V rocket.*

- Click on the *Centrifugal Pump* devices and navigate to the *Centrifugal Pump Results* heading in the *Property Grid*. As seen in Figure 10.9, the *Power* required by each turbopump volute is listed:
  - For the fuel volute, the required power is 22,615 hp; and
  - For the oxidizer volute, the required power is 30,683 hp.

Summing the required power values for each turbopump volute, we reach the conclusion that the *total power requirement* for each F-1 turbopump is **53,298 hp**. It is obvious that each F-1 engine requires an immensely powerful pump! Now, NASA reports the power requirement for each F-1 turbopump as 52,926 hp. Impressively, our simulated value deviates less than 1% from the actual value. While using PIPE-FLO may not be rocket science, even rocket scientists can benefit from a strong knowledge of fluid mechanics!


(A)

Property	Value
Centrifugal Pump Design	
Centrifugal Pump Results	
Flow Rate	15632 gpm
Total Head	5224 ft
Differential Total Pressure	1803 psi
Differential Static Pressure	1652 psi
NPSHa	109.8 ft
Suction Total Pressure	38.1 psi a
Inlet Static Pressure	27.33 psi a
Discharge Total Pressure	1841 psi a
Outlet Static Pressure	1679 psi a
Speed	5500 rpm
Input Power	22615 hp
Output Power	16433 hp
Efficiency	72.62 %
NPSHr	--
% Deviation	0 %

(B)

Property	Value
Centrifugal Pump Design	
Centrifugal Pump Results	
Flow Rate	25038 gpm
Total Head	3172 ft
Differential Total Pressure	1568 psi
Differential Static Pressure	1568 psi
NPSHa	99.44 ft
Suction Total Pressure	64.14 psi a
Inlet Static Pressure	54.23 psi a
Discharge Total Pressure	1632 psi a
Outlet Static Pressure	1622 psi a
Speed	5500 rpm
Input Power	30683 hp
Output Power	22895 hp
Efficiency	74.55 %
NPSHr	--
% Deviation	0 %

Figure 10.9: *The simulated power required to operate the F-1 turbopump **(A)** fuel and **(B)** oxidizer volutes.*

Tutorial Problems

In lieu of the regular tutorial activities, consider the follow questions related to the turbopump simulation developed in this tutorial:

1. Pipes 1, 2 & 4 have a *negative pressure drop*. What is causing this phenomenon?
2. Which pipes have the highest velocities of fluid? Why is this?
3. Why is it undesirable to use *Flow Demand* devices instead of *Control Valves* and *Pressure Boundaries*?
4. If the ambient conditions at launch time were varied, the temperature and density of the RP-1 fuel would also change. What would be the effect of higher/lower RP-1 density on the operation of the system?
5. Why was important that the oxidizer tank was maintained at a temperature of -297°F?
6. As the rocket burned fuel and oxidizer, the liquid levels in both tanks decreased. What is the effect this decrease in liquid level on the operation of the propellant feed system?

Works Cited

J.C. Oefelein and V. Yang. *Comprehensive Review of Liquid-Propellant Combustion Instabilities in F-1 Engines.* J. Propul. Power, 9 (1993) 5, pp. 657–678.

F-1 Rocket Engine Technical Manual R-3896-1. National Aeronautics and Space Administration, March 1967. Accessible: latulippelab.mcmaster.ca/F-1_Rocket_Engine_Test_Manual.pdf or http://www.christophermadsen.com/docs/docs.htm.

Liquid Rocket Lines, Bellows, Flexible Hoses, and Filters. National Aeronautics and Space Administration, April 1977. Accessible: https://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/19780008146.pdf.

Saturn V Flight Manual SA 503. National Aeronautics and Space Administration, November 1968. Accessible: https://history.nasa.gov/afj/ap08fj/pdf/sa503-flightmanual.pdf.

Saturn V News Reference. National Aeronautics and Space Administration, December 1968. Accessible: https://history.msfc.nasa.gov/saturn_apollo/documents/F-1_Engine.pdf.

J.W. Magee *et al.* *Thermophysical Properties Measurements and Models for Rocket Propellant RP-1: Phase I.* National Institute of Standards and Technology, February 2007. Accessible: http://nvlpubs.nist.gov/nistpubs/Legacy/IR/nistir6646.pdf.

Turbopump Systems for Liquid Rocket Engines. National Aeronautics and Space Administration, August 1974. Accessible: https://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/19750012398.pdf.

N.F. Dubovkin *et al.* *Vapor Pressure and Critical Parameters of Jet Fuel.* Plenum Publishing Corporation (1981) pp. 207–210.

Acknowledgements

Figure 10.1: The Saturn V Rocket (Image)
NASA
NASA Image Galleries, NASA
© National Aeronautics and Space Administration (NASA)
Reprinted with permission.

Figure 10.2: The Saturn V Rocket Schematic (Stage S-IC)
NASA
NASA History, NASA
© National Aeronautics and Space Administration (NASA)
Reprinted with permission.

Figure 10.3: Schematic of S-IC (with F-1 engine highlighted)
NASA
Saturn V Flight Manual SA 503, NASA
© 1968 National Aeronautics and Space Administration (NASA)
Reprinted with permission.

Figure 10.4: A schematic of the Rocketdyne F-1 engine
NASA
Saturn V News Reference, The Boeing Company, McDonnel Douglas Astronautics Company, Space Division of North American Rockwell Corporation, Rocketdyne Division of North American Rockwell Corporation, and IBM Corp in cooperation with NASA
© 1967 National Aeronautics and Space Administration (NASA)
Reprinted with permission.

Figure 10.7: Actual pump curves (head vs. flow rate) for the Rocketdyne F-1 turbopump
NASA
F-1 Rocket Engine Technical Manual R-3896-1, NASA
© 1967 National Aeronautics and Space Administration (NASA)
Reprinted with permission.

End of Tutorial