Using CHEMCAD for Piping Network Design and Analysis  
Part 1 – Unbranched Pipes  

By  
Jeffrey L. Silverstein, PhD  
Director  
Process Simulation Express LLC  
Ramsey, NJ 07446  

Abstract  
CHEMCAD is a powerful tool for designing and analyzing complex piping networks. It enables the engineer to integrate the piping network simulation into the overall process simulation, calculating a momentum balance in addition to a heat and material balance. This makes it possible to quickly evaluate the effects of changes in the process concept, or of its chemicals, on the piping network.  

Part 1 of this paper presents a tutorial on how to use the piping network tool for unbranched pipe analysis. Part 2 progressively adds complexity with more complicated networks, culminating in examples that demonstrate how CHEMCAD can be used to analyze practical plant piping problems.  

Introduction  
The CHEMCAD software suite is a powerful tool for designing and analyzing complex piping networks (1). As shown in Figure 1, CHEMCAD contains a number of UnitOps that calculate flow as a function of pressure drop. These are: Pipe Simulator, Control Valve, Valve, Pump, Node, Compressor, and Expander.  

![CHEMCAD UnitOps that calculate flow as a function of pressure drop](image)

Since this paper focuses on liquid piping systems, we will examine the first five of these unit operations in CHEMCAD to show how they can be combined to simulate complex piping networks.
We will begin by examining the Pipe Simulator and showing how it can be used for pipe sizing. We will explore the various calculation options available for this UnitOp, and add a pump to construct a simple single-line example. We will review the features of the Pump UnitOp, and see how the Sensitivity Study feature of CHEMCAD can be used to obtain pump head vs. flow rate for the system. Then we will add a pump curve to characterize the pump, and calculate the actual flow rate through the system.

To achieve a desired flow rate, we will add a control valve to the system, and show how the control valve sizing option in CHEMCAD can be used as a starting point for actual valve selection.

We will also add a flow restriction orifice to the system using CHEMCAD, and determine the effect on the valve position to accommodate this new flow resistance.

In Part 2, we will examine a more complicated two-branch flow problem. The significantly more complex calculation method necessary to solve this problem will be evaluated in some detail. We will then introduce a nodal approach to solving this problem, and demonstrate the improved calculation efficiency achieved. Two other basic branched problems will be solved to demonstrate the utility and power of the nodal approach.

We will follow up with practical plant piping examples, which can be analyzed in CHEMCAD to understand and correct the plant issue.

It is recommended that the reader download, open, and refer to referenced CHEMCAD files while reading about the approaches to solving the various problems discussed in this paper.

**Pipe Simulator UnitOp**

Figure 2 shows the CHEMCAD flowsheet for a simple pipe. The filename is #1SizePipe. The fluid is water and the flow rate is 20 gpm. The pipe has a 200' horizontal run 2-90° elbows, and a 200' vertical run with one ball valve and 3-90° elbows. What pipe size should we use?

![Pipe Simulator UnitOp](image)

Three Variables: F1,P1,P2,

One Equations:

\[ F1 = |P1, P2| \]

Must specify 2 variables to constrain the system: \( F = 20 \text{ gpm water at 60F}; P1 = 200 \text{ psia} \)

Piping Run: 1 horizontal run of 200 ft with 2 - 90 degree elbows, 1 vertical run of 200 ft with one ball valve and 3 - 90 degree elbows

*Figure 2 – Simple piping case – pipe sizing*
CHEMCAD provides a pipe sizing tool, found under Sizing on the main menu. To use it, you must first specify the stream (stream 1) that will flow through the pipe. To do this, double-click the stream line to open the Edit Streams dialog, shown in Figure 3.

![Figure 3 – The Edit Streams dialog box for stream 1](image)

As you can see, the stream temperature, pressure, and flow rate in gpm are specified as inputs. The vapor volume fraction is calculated. Only two of these three parameters can be specified; the third is calculated by CHEMCAD. To specify the total flow in gpm, it is necessary to specify the component unit in compositional units. If mass units are specified, the total flow cannot be specified in volume units.

Now right-click stream 1 and select Sizing > Line Sizing, (see Figure 4) then click OK in the Select Streams dialog box. This brings up the Pipe Sizing dialog box, shown in Figure 5. Note that you can also navigate to this dialog by selecting Sizing > Piping from the main menu.

![Figure 4 – Selecting Line Sizing from the right-click menu](image)
Three sizing options are offered: Typical sizing for single phase, Sizing based on frictional Dp/100 ft, and Sizing based on velocity. Note that the pipe schedule must be specified, but that the default for pipe roughness is for steel pipe. Either of these entries can be changed if desired. There are two choices of flow method, with single phase and Baker’s two-phase method being the first and Beggs and Brill’s two-phase method being the second. Since this is a single-phase flow problem, choose the first method.

For this example, use Typical sizing for single phase. When you click OK, the results shown in Figure 6 appear in a new tab.

CHEMCAD suggests a pipe size of 1.5 in., and shows the result for one pipe size larger, 2.0 in. and one pipe size smaller, 1.0 in. How accurate are the calculated pressure drops in psi/100 ft for each pipe size? Table 2-7, Flow of Water Through Schedule 40 Steel Pipe (2), provided at the end of this paper, gives the answer. Comparison results are given in Figure 7. The CHEMCAD and Crane frictional pressure drops match up very well. So, we can have confidence in the CHEMCAD results.
Example 1. Simple Piping Case - Pipe Sizing

Comparing CHEMCAD Results To Technical Paper No. 410, Crane Co., Engineering Div., Chicago (1957)

- Liquid only -

<table>
<thead>
<tr>
<th></th>
<th>CHEMCAD</th>
<th>Crane Tech Paper 410</th>
<th>CHEMCAD</th>
<th>Crane Tech Paper 410</th>
<th>CHEMCAD</th>
<th>Crane Tech Paper 410</th>
</tr>
</thead>
<tbody>
<tr>
<td>Schedule</td>
<td>40</td>
<td>40</td>
<td>40</td>
<td>40</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flow Regime</td>
<td>Single phase</td>
<td>Single phase</td>
<td>Single phase</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pipe ID in</td>
<td>1.61</td>
<td>2.065</td>
<td>1.049</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Nominal Dia. in</td>
<td>1.5</td>
<td>2</td>
<td>1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Press Drop psi/100ft</td>
<td>1.2916</td>
<td>1.28</td>
<td>0.3791</td>
<td>0.375</td>
<td>11.0342</td>
<td>10.9</td>
</tr>
<tr>
<td>Velocity ft/sec</td>
<td>3.1543</td>
<td>3.16</td>
<td>1.9174</td>
<td>1.91</td>
<td>7.4302</td>
<td>7.43</td>
</tr>
</tbody>
</table>

*Figure 7 – CHEMCAD results compared to Crane*

How should you choose what pipe diameter to use? Let’s look at the fluid velocity for each pipe size.

Velocities for various fluids are given in Table 2-2, Suggested Fluid Velocities in Pipe and Tubing, (Ludwig, 1977). The table suggests water velocities in the range 3 - 8 ft/sec, with the caveat that “The velocities are suggestive only and are to be used to approximate line size as a starting point for pressure drop calculations. The final line size should be such as to give an economical balance between pressure drop and reasonable velocity.”

The velocities for the pipe sizes CHEMCAD offers in Figure 6 are in the suggested range for the 1.0 in. and 1.5 in. diameter pipe. Based on Table 2-2, the 2.0 in. pipe diameter would be rejected. The choice between the 1.0 and 1.5 in. pipe diameters would be based on the cost of piping, fittings, valves, and the pump, with the smaller pipe size likely being more economical.

The third option for pipe sizing given in Figure 5 is Sizing based on velocity. If you choose this option, the dialog box provides an input for velocity (see Figure 8); if you use 6 ft/sec as the desired velocity, you get the results shown in Figure 9.

*Figure 8 – Pipe Sizing dialog with Velocity field enabled*
These results are identical to those obtained by letting CHEMCAD determine typical sizing. This is not surprising, because pipes are available only in discrete sizes. The flow conditions have to conform to what is commercially available. CHEMCAD gives the minimum pipe diameter that does not exceed the velocity specification.

The second option for pipe sizing is Sizing based on \( \frac{Dp}{100 \text{ ft}} \). I typically think of this as a result of sizing a pipe, rather than an input. The first and third sizing options are sufficient for most situations.

Let’s use 1.5 in Schedule 40 pipe and enter the required information in the Pipe UnitOp. Double-click the Pipe UnitOp to open the Pipe Sizing and Rating dialog box. As shown in Figure 10, a variety of flow options are offered in the Method drop-down box. Figure 11 shows the various Sizing options.
For now, choose **Single Phase flow** as the method and **Design, single phase flow** as the sizing option, and leave the input for pipe diameter blank, as shown in **Figure 12**.

Although the pipe length and elevation changes are specified here, you can obtain a pipe diameter with only the feed stream flow information, the pipe schedule, and the pipe roughness.

Click **OK**, and then right-click UnitOp 1 and select **Run This UnitOp**. After the unit runs, you can reopen the pipe dialog and see that the pipe diameter chosen is 1.5-inch Schedule 40. The flow calculation is also made through the pipe as configured in the dialog box.

Before we look at the calculated results, let’s again check the inputs for the pipe configuration. The pipe diameter, pipe schedule, pipe MOC and roughness, friction factor calculation method, total pipe length, and pipe vertical elevation are all specified. Two other pipe diameter cases can be specified, as I will discuss later. The use of pipe segments and fluid holdup in the pipe are explained in the help topic for the Pipe UnitOp.
Clicking on the Valves and Fittings tabs shows the available choices and the selections made for this example. These are shown in Figures 13 and 14, respectively. Clicking on the Properties tab shows the physical properties of the fluid used for the pipe calculations, which is water in this case.

![Figure 14 – The Pipe UnitOp’s Valves tab](image1)

![Figure 13 - The Pipe UnitOp’s Fittings tab](image2)

Let’s see what CHEMCAD calculates, as shown by clicking on the Calculated Results tab and given in Figure 15.

![Figure 15 – The Pipe UnitOp’s Calculated Results tab](image3)
The Delta P for elevation and the frictional Delta P are given as separate items. The pressure drop at the top of the list is simply the sum of the two, 91.99 psi. The pressure drop/100 ft is calculated from the frictional pressure drop and the total equivalent length of pipe, which includes the effect of fittings and valves. The velocity, Reynolds Number, and friction factor are also given. If you select Report > UnitOps > Select UnitOps from the main menu, you can obtain the same information and a bit more, in a form that you can print if desired.

Before moving on, let’s take a look at the sizing options under the Specifications tab, as shown in Figure 11:

- If the 0 option, Rating, is chosen, the pipe outlet pressure is calculated when the pipe diameter, the flow rate, and the inlet pressure are specified. Running with these options produces identical results to those given in Figure 14, where Delta P = 91.99 psi.
- Options 1, 2, and 6 are the design options, discussed above. Option 3 is for two-phase flow and will not be discussed in this paper.
- Option 4 is the reverse of the rating option: the outlet pressure is specified and the inlet pressure is calculated. Let’s choose this option and set the inlet and outlet pressures to 14.7 psia. Since the flow rate is unchanged, the delta P should also be unchanged at 91.99 psi. If you run this case, the inlet pressure is 106.69 psia and the delta P is indeed 91.99 psia, as expected.
- Option 5 calculates the flow rate with the pipe diameter and the inlet and outlet pressures as inputs. Try using this option and increasing the inlet pressure from the current 106.69 psia to 164.7 psia. Keep the outlet pressure at 14.7 psia so that Delta P = 150 psia. The flow rate increases to 73.6 gpm. Option 5 is important for more complicated branched networks, as will be discussed in Part 2 of this paper.

Now reset the flow rate and pressure of stream 1 to 20 gpm and 200 psia, respectively. Select option 0 and specify the diameter as 1.5 inches. In addition, let’s do a case study and specify two other pipe diameters in the dialog box—1.0 inch and 2.0 inch—as shown in Figure 16.

![Figure 16 – The Pipe UnitOp’s Calculated Results tab](image)

If you run the pipe simulation and then look at the Calculated Results tab, you’ll see that the results are identical to what you saw in Figure 6, as expected.
Add a Pump

Now let’s add a pump to the problem, as shown in Figure 17. The filename is #2SimpleCaseWithPump.

![Pump Diagram]

**Piping Run after Pump:** 1 horizontal run of 200 ft. with 2 - 90 degree elbows  
1 vertical run of 200 ft. with one ball valve and 3 - 90 degree elbows

What is the outlet pressure of the pump corresponding to P2 = 14.7 psia?

*Figure 17 – Single pipe with pump*

In this example the water flow rate is 200 gpm. The stream information for stream 1 is given in Figure 18.

*Figure 18 – Specifications for stream 1*

If we use the typical pipe sizing option for stream 1, we obtain a pipe size of 4 inches with a fluid velocity of 5 ft/sec, within the suggested range of 3 – 8 ft/sec. Typical guidelines for suction side piping for centrifugal pumps often recommend the same pipe size as the outlet piping or one pipe size larger. In this example, we will use 6 inches as the suction side pipe size, and 4 inches as the downstream pipe size. We will need to provide a reducer on the suction side of the pump at the pump inlet nozzle. In addition, there should be at least 5 – 10 pipe diameters of straight pipe entering the pump to ensure optimal suction.

*Figure 19* shows the suction side pipe dialog box. It is 6 in. Sch 40, 20 feet long with a drop in elevation of 8 feet. There are 2 ball valves in this line, one reducer and two 90° elbows. The pump outlet line is 4 in. Sch 40 and is 400 feet long with an increase in elevation of 200 feet. It has one ball valve and five 90° elbows.
What is the required pressure at the pump outlet to deliver this flow? Figure 20 shows the pump dialog box.

The pump can be on or off. There is a choice of modes or calculation basis. The pump outlet pressure, the pressure increase across the pump, or the pump curve can be specified. In this example, we will specify the pump outlet pressure as 150 psia. In later examples, we’ll explore the more typical situation when the pump curve is known. We can also request that CHEMCAD calculate the available NPSH; we’ll cover this in more detail later.

After the simulation runs, the flow rate of 200 gpm set for stream 1 is unchanged. As shown in Figure 21, the pump outlet pressure is 150 psia, as specified and the outlet pressure of the system is 59 psia.
If the desired pressure is 14.7 psia, how can that be achieved? One simple way is to recognize that the pressure drop across pipe element #1 for a flow rate of 200 gpm is fixed. This pressure drop is 150 - 59 = 91 psi. The pump pressure must be 14.7 + 91 = 105.7 psia. If we set this as the new pump pressure and run the simulation, the desired pipe outlet pressure will be achieved, as shown in Figure 22.

**Figure 21 – Pipe 1 inlet and outlet streams**

| Stream No. | 8   | 2   |
| Stream Name | PUMP P | P2 |
| Temp F     | 60.6507 | 60.6507 |
| Pres psia  | 180.0000 | 59.9982 |
| Enthalpy kBtu/h | -6.843E+005 | -6.843E+005 |
| Vapor mole frac. | 0.00000 | 0.00000 |
| Total lbmol/h | 5558.9629 | 5558.9629 |
| Total lb/h   | 100144.7109 | 100144.7109 |
| Total std L gpm | 200.0000 | 200.0000 |
| Total std V scfh | 2109507.50 | 2109507.50 |
| Flow rates in lb/h | 100144.7109 | 100144.7109 |
| Water       | &nbsp;  | &nbsp; |

**Figure 22 – Pipe 1 inlet and outlet streams with new pump pressure**

**Add a Controller**

A more efficient way to determine the required pump pressure is by adding a feedback controller to the flowsheet as in Figure 23. The filename is #3SimpleCaseWithPump&Controller. In this way, any other changes in the system like flow rate or piping configuration, will be automatically accounted for without the need for a hand calculation.

**Figure 23 – Adding a feedback controller to the flowsheet**

Add a Pump and Controller
The controller sets the pump outlet pressure so that the required P2 is obtained
Generate System Operating Curve

Piping Run: 1 horizontal run of 200 ft, 2 in Sch 40 pipe with 2 - 90 degree elbows
1 vertical run of 200 ft with one ball valve and 3 - 90 degree elbows
The dialog box for the controller is given in Figure 24. We are determining the pump outlet pressure for a desired system outlet pressure of 14.7 psia. Under the Feedback Options tab, set the tolerance to 1e-5 and let CHEMCAD decide how to vary the pump outlet pressure to best arrive at the desired outlet pressure.

![Controller settings](image)

**Figure 24 – Controller settings**

Before running the simulation, let’s arbitrarily set the pump outlet pressure at 200 psia and run the simulation with the controller turned off. Set the Controller Mode to Controller off. The results of the simulation are given in Figure 25, for all streams in the flowsheet.

```
Simulation: #SSimpleCaseWithPumpsController
FLOW SUMMARIES:

<table>
<thead>
<tr>
<th>Stream No.</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stream Name</td>
<td>P1</td>
<td>F2</td>
<td>SYS OUT</td>
<td>PUMP P</td>
</tr>
<tr>
<td>Temp F</td>
<td>60.0000*</td>
<td>60.5398</td>
<td>60.5398</td>
<td>60.5398</td>
</tr>
<tr>
<td>Press psia</td>
<td>14.6960*</td>
<td>108.9965</td>
<td>108.9965</td>
<td>200.0000</td>
</tr>
<tr>
<td>Enthalpy kbtu/h</td>
<td>-6.844E+005</td>
<td>-6.844E+005</td>
<td>-6.844E+005</td>
<td>-6.844E+005</td>
</tr>
<tr>
<td>Mole fraction</td>
<td>0.000000</td>
<td>0.000000</td>
<td>0.000000</td>
<td>0.000000</td>
</tr>
<tr>
<td>Total lbmol/h</td>
<td>5558.9629</td>
<td>5558.9629</td>
<td>5558.9629</td>
<td>5558.9629</td>
</tr>
<tr>
<td>Total lb/h</td>
<td>100144.7109</td>
<td>100144.7109</td>
<td>100144.7109</td>
<td>100144.7109</td>
</tr>
<tr>
<td>Total std L gpm</td>
<td>200.0000</td>
<td>200.0000</td>
<td>200.0000</td>
<td>200.0000</td>
</tr>
<tr>
<td>Flow rate in lb/h</td>
<td>2109507.50</td>
<td>2109507.50</td>
<td>2109507.50</td>
<td>2109507.50</td>
</tr>
<tr>
<td>Water</td>
<td>100144.7109</td>
<td>100144.7109</td>
<td>100144.7109</td>
<td>100144.7109</td>
</tr>
</tbody>
</table>
```

![Stream flow summaries with controller off](image)

**Figure 25 - Stream flow summaries with controller off**
The Delta P across the downstream pipe is 91 psi as we previously calculated. The system outlet pressure is 109 psia. The pump outlet pressure of 200 psia is set. Note that the stream pressure at the pump inlet, 18 psia, is higher than the pressure at the upstream pipe inlet of 14.7 psia. CHEMCAD accounts for the hydrostatic pressure increase due to the 8-foot drop in elevation.

Now let’s turn the feedback controller on and run the simulation to determine the pump pressure corresponding to the system outlet pressure of 14.7 psia. Figure 26 shows the simulation results and that the pump outlet pressure is correctly determined at 105.7 psia.

FLOW SUMMARIES:

<table>
<thead>
<tr>
<th>Stream No.</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>8</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stream Name</td>
<td>P1</td>
<td>P2</td>
<td>SYS OUT</td>
<td>PUMP P</td>
</tr>
<tr>
<td>Temp F</td>
<td>60.0000</td>
<td>60.2600</td>
<td>60.2600</td>
<td>60.2600</td>
</tr>
<tr>
<td>Enthalpy kBTU/h</td>
<td>-6.8449E+005</td>
<td>-6.8447E+005</td>
<td>-6.8447E+005</td>
<td>-6.8447E+005</td>
</tr>
<tr>
<td>Vapor mole frac.</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>Total lbmol/h</td>
<td>5558.9629</td>
<td>5558.9629</td>
<td>5558.9629</td>
<td>5558.9629</td>
</tr>
<tr>
<td>Total lb/h</td>
<td>100144.7109</td>
<td>100144.7109</td>
<td>100144.7109</td>
<td>100144.7109</td>
</tr>
<tr>
<td>Total std L gpm</td>
<td>200.0000</td>
<td>200.0000</td>
<td>200.0000</td>
<td>200.0000</td>
</tr>
<tr>
<td>Total std V scfh</td>
<td>2109507.50</td>
<td>2109507.50</td>
<td>2109507.50</td>
<td>2109507.50</td>
</tr>
<tr>
<td>Flow rates in lb/h</td>
<td>100144.7109</td>
<td>100144.7109</td>
<td>100144.7109</td>
<td>100144.7109</td>
</tr>
</tbody>
</table>

Figure 26 – Stream flow summaries with controller on

This flowsheet can be used to determine the operating curve for this system. This curve shows the relationship between the pump outlet pressure, or head, and the system flow rate. In CHEMCAD, this can easily be calculated using the powerful Sensitivity Study feature. With a sensitivity analysis, we can make repetitive simulations automatically changing the flow rate incrementally and calculating the pump head at each increment. You can generate a graph of pump head vs. flow rate that can be superimposed on the pump operating curve to determine the pump’s operating point and actual system flow rate.
To access an existing sensitivity analysis, select **Run > Sensitivity Study > [study name] > Edit**, as shown in **Figure 27**. The name of the analysis in this case is *Pump Head vs Flow Rate*.

**Figure 27 – Accessing the sensitivity study**

The first tab in the Edit Sensitivity Study dialog box is called **Adjusting**, as shown in **Figure 28**. On this page we specify the independent variable we wish to very incrementally.

**Figure 28 – The Adjusting tab**

The independent variable is the flow rate of Stream 1 in gpm. Specify the flow range you want and the size of each step change in flow. The flow range is 50 – 500 gpm in 25 gpm increments. Sensitivity analysis in CHEMCAD allows us to specify a second independent variable that will provide parametric plots. This can be extremely useful in applications where we want to examine the effect of changing two system variables, for example, temperature and pressure. A second independent variable is not needed in this example.

The next three tabs of the dialog box are labelled **Recording**. As shown in **Figures 29 and 30**, these pages allow you to record up to twelve variables of interest for the analysis at each incremental step of CHEMCAD’s calculations.
With the variables all specified, you can now execute the **Run All** command for the sensitivity study, using the menu path shown in **Figure 27**. Tabular results can be obtained by clicking on **Report Results** in the same submenu. The tabulated results are given in **Figure 31**.
Figure 31 – Tabular results from the sensitivity study

To obtain graphical results, use the same menu path and click on **Plot Results**. A new dialog box opens as shown in **Figure 32**, giving the variables you might want to plot. In this case you should plot Head vs system flow rate. The graph produced is given in **Figure 33**.

![Sensitivity Plot dialog box](image)
**Figure 33 – Sensitivity study plot**

**Add the Pump Curves**

Typical pump operating curves are shown in **Figure 34 (4).**
This paper by Fernandez, K., et al., entitled “Understanding the Basics of Centrifugal Pump Operation,” provides an excellent summary of the topic. The pump operating curves relate total head, power, NPSHR, and efficiency to flow rate for specific pump impeller diameters. Total head is defined as the difference in pressure between the outlet and inlet of the pump expressed in feet of water column.

The flowsheet for adding pump curves is shown in Figure 35. The filename is #4SimpleCaseWithPumpCurve&Controller.

Add a Pump and Controller
Use Pump Curve
The controller sets the flow rate of stream 1 so that the required P2 is obtained

Concepts Demonstrated
1. Using a Pump Curve to characterize the pump’s operation
2. Obtain the true operating point of the pump for this system - where the pump curve and the system operating line cross

Figure 35 – Flowsheet for pump curve

If you open the pump dialog box and choose Specify performance curve, as shown in Figure 36, you can see that the pump speed is a required specification. Pump speed should be available from the pump vendor and is often provided on the pump curves. Clicking OK produces the filled-in input form for pump curves, as shown in Figure 37.
We can plot the total head vs flow rate curve for the pump from Figure 37 and the head vs flow rate for the system from the table given in Figure 31 on an Excel spreadsheet, as shown in Figure 38. Where they cross is the flow rate the pump should deliver. This flow rate is about 366 gpm.
Now let’s run the CHEMCAD simulation for the flowsheet in Figure 35 to see if this result is matched. Figure 39, obtained by clicking Report > Stream Properties > All Streams on the main CHEMCAD menu, shows that the flow rate is indeed accurately determined by CHEMCAD using the pump curve. If a lower flow rate is desired, say 200 gpm, a control valve needs to be added to the line.

![Figure 38 – System operating line and pump curve](image)

### NPSHR and NPSHA

However, before we do this, let’s digress and define NPSHR, net positive suction head required, and NPSHA, net positive suction head available. The former is a characteristic of the pump and must be specified by the pump manufacturer. The latter is a characteristic of the pipe configuration and the fluid being pumped. It must be calculated as is done by CHEMCAD.

NPSHA is the absolute pressure at the suction port of the pump.
NPSHR is the minimum pressure required at the suction port of the pump to prevent the pump from cavitating.

To prevent cavitation, NPSHA must be greater than NPSHR. More suction side pressure must be available than is required by the pump. Pump cavitation occurs when the pressure at the pump inlet is below the vapor pressure of the liquid. Vapor bubbles form at the pump inlet and are moved to the discharge side of the pump where they collapse often causing pitting and damage to the pump impellers. If cavitation occurs, the pump will be noisy, will experience loss of capacity and reduced operational life. NPSHA is calculated from the following equation:

\[
\text{NPSHA} = \text{HA} + \text{HZ} - \text{HF} + \text{HV} - \text{HVP}
\]

where:

HA = the absolute pressure on the surface of the liquid in the supply tank.

HZ = the vertical distance between the surface of the liquid in the supply tank and the centerline of the pump. This is positive if the supply tank is above the pump and negative if it is below. As the feed tank level drops during pumping, the NPSHA will also drop.

HF = the frictional losses in the suction piping from the supply tank to the pump inlet.

HV = the velocity head at the pump suction port. This is usually small and is often ignored.

HVP = absolute vapor pressure of the liquid at the pumping temperature.

The calculations should be done in pressure units of feet of water column to match the way NPSHR is usually specified. In particular, HZ, the liquid static head must take liquid density into account for liquids other than water.

For the example of Figure 35, NPSHA is calculated as follows using results obtained from the CHEMCAD simulation:

\[
\text{NPSHA} = 14.696 + 3.4682 - 0.2802 + 0 - 0.2564 = 17.383 \text{ psia}
\]

This is equivalent to 40.66 feet of water column. CHEMCAD calculates 40.7 feet as shown in Figure 40, obtained from Report > UnitOps > Select UnitOps > 6 > OK from the main CHEMCAD menu.

**Figure 40 – Pump summary**
Add a Control Valve to Achieve a Desired Flow Rate

Now let’s add a control valve downstream of the pump to control the flow to the desired 200 gpm. There are many online sources for control valve sizing (5-8). Blackmonk Engineering (7) in particular, provides an excellent step-by-step calculation guide for sizing a control valve. Guidelines for choosing a linear valve or an equal percentage valve are discussed in (9).

CHEMCAD provides a control valve sizing option. It is important to point out that this is a starting point for actual valve selection. I recommend that you use a sizing guide similar to (7) and vendor-supplied software to finalize the design.

To begin the control valve sizing for this problem, estimate the required control valve pressure drop to achieve 200 gpm flow rate. We can easily do this by going back to the simulation with filename #3SimpleCaseWithPump&Controller. This flowsheet corresponds to the case where we set the inlet flow to 200 gpm and used a controller to set the outlet pump pressure so that the system outlet pressure is 14.7 psia.

As shown in Figure 41, for simulation with filename #5SimpleCaseEstimateControlValveDeltaP, we add a control valve and show the pressure at various points in the pipe line corresponding to 200 gpm flow rate. Note that the calculation controller is removed because the control valve outlet pressure can be set in the control valve dialog box.

**Estimate Required Delta P Across Control Valve For Flow = 200 gpm Water**

![Diagram of control valve setup](Image)

Piping Run after Pump: 1 horizontal run of 200 ft, with 2 - 90 degree elbows
1 vertical run of 200 ft with one ball valve and 3 - 90 degree elbows

**Figure 41 – Flowsheet for estimating control valve delta P**

The actual pump outlet pressure is obtained from the total head vs flow rate curve for the pump. Remember that the total head is the difference between the pump outlet and inlet pressures expressed in units of feet of water column. The total head at 200 gpm is 327.8 feet of water column, as shown in Figure 37. This corresponds to 142.08 psia. Adding this to the pump inlet pressure of 18.08 psia gives an absolute pump outlet pressure of 160.19 psia. CHEMCAD calculates 159.97 psia, which we will use in our calculations.

The pressure drop across the downstream pipe for 200 gpm flow is 91 psia, from the simulation #3SimpleCaseWithPump&Controller. Thus, the pressure at the pipe outlet is 68.97 psia. We know that the system outlet pressure is 14.7 psia, so the control valve pressure drop must be approximately 54.27 psia.
Now let’s size the control valve with CHEMCAD in #5SimpleCaseEstimateControlValveDelta P. We need to determine the valve inlet pressure, so we set the outlet pressure of the pump to 159.97 psia, as shown in the dialog box in Figure 42.

![Figure 42 – Pump dialog for control valve sizing](image)

Figure 43 shows the unspecified control valve dialog box. As you can see, the valve Cv and downstream pressure are unspecified. The defaults are linear valve and calculate valve position for a specified flow rate.

![Figure 43 – Unspecified Control Valve dialog box](image)

We now run the simulation for the units upstream of the valve, namely, the pump inlet pipe, the pump, and the outlet pipe. This establishes the inlet pressure to the control valve. The stream summary in Figure 44 shows that this is 68.97 psia.
If we now click on Stream 2, the control valve inlet stream, we can select control valve sizing under the sizing option. This opens the dialog box in Figure 45, which is filled in for this example; the outlet pressure is set at 14.7 psia.

Clicking OK brings up the sizing recommendation given in Figure 46. The stream pressure information we provided is used in the valve sizing algorithm.

CHEMCAD recommends a 2-inch valve with a corresponding Cv of 36. The calculated Cv is what is required for the problem and CHEMCAD chooses the next higher number corresponding to a valve size, in this case 36.
Please note that CV and valve size may vary from vendor to vendor. For example, an Emerson 2-inch ES globe valve with linear trim has a CV of 65.3 at a valve opening of 100% (7). Returning to the valve dialog box by double-clicking the valve symbol, we see that the valve CV and valve outlet pressure are entered automatically.

Now let’s run the full flowsheet. Figure 47 provides a stream summary showing all of the stream pressures as indicated on the flowsheet.

**FLOW SUMMARIES: CONTROL VALVE SIZING**

<table>
<thead>
<tr>
<th>Stream No.</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>8</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stream Name</td>
<td>14.7 psia</td>
<td>68.97 psia</td>
<td>14.7 psia</td>
<td>159.97 psia</td>
</tr>
<tr>
<td>Temp F</td>
<td>60.0000</td>
<td>60.6590</td>
<td>60.6592</td>
<td>60.6594</td>
</tr>
<tr>
<td>Pres psia</td>
<td>14.7000</td>
<td>68.9701</td>
<td>14.7000</td>
<td>159.9718</td>
</tr>
<tr>
<td>Enth kBTU/h</td>
<td>-6.8449E+005</td>
<td>-6.8449E+005</td>
<td>-6.8449E+005</td>
<td>-6.8449E+005</td>
</tr>
<tr>
<td>Vapor mole frac.</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>Total lbmol/h</td>
<td>5558.9629</td>
<td>5558.9629</td>
<td>5558.9629</td>
<td>5558.9629</td>
</tr>
<tr>
<td>Total lb/h</td>
<td>100144.7109</td>
<td>100144.7109</td>
<td>100144.7109</td>
<td>100144.7109</td>
</tr>
<tr>
<td>Total std L gpm</td>
<td>200.0000</td>
<td>200.0000</td>
<td>200.0000</td>
<td>200.0000</td>
</tr>
<tr>
<td>Total std V scfh</td>
<td>2109507.50</td>
<td>2109507.50</td>
<td>2109507.50</td>
<td>2109507.50</td>
</tr>
<tr>
<td>Flow rates in lb/h</td>
<td>100144.7109</td>
<td>100144.7109</td>
<td>100144.7109</td>
<td>100144.7109</td>
</tr>
<tr>
<td>Water</td>
<td>100144.7109</td>
<td>100144.7109</td>
<td>100144.7109</td>
<td>100144.7109</td>
</tr>
</tbody>
</table>

**Figure 47 – Summary showing all stream pressures in flowsheet**

Opening the valve dialog box shows that the valve is 84% open. If we try a 3-inch valve with CV of 75 as per CHEMCAD, the valve is 40.3% open. The usual guideline for valve opening at maximum expected flow rate is 20-80%. Accordingly, I would choose the 3-inch control valve. The valve can also be positioned between the pump and the pipe as shown in the file called #6SimpleCaseControlValveAfterPump.

**Add a Restriction Orifice**

CHEMCAD permits the addition of a restriction orifice plate to a piping system. This is demonstrated in the CHEMCAD file called #7SimpleCaseWithControlValve&RO. To size the RO, right-click stream 4, the pump outlet, and select Sizing > Orifice Sizing. This opens the filled-in orifice sizing dialog box, as shown in Figure 48.

**Figure 48 – The Orifice Sizing dialog box**
Required inputs are pressure tap locations, pipe inside diameter, and pressure drop in inches of water column. Information about orifice configuration and pressure tap locations is available in the Chemical Engineers’ Handbook (10). The pressure drop chosen for this example is 10 psi. If you click OK, the CHEMCAD recommended sizing is shown, as in Figure 49.

**Orifice Sizing for Stream # 4**

<table>
<thead>
<tr>
<th>Loadings and Properties</th>
<th>Vapor</th>
<th>Liquid</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow rate</td>
<td>0.0000 lb/h</td>
<td>100144.7109 lb/h</td>
</tr>
<tr>
<td>Flow rate</td>
<td>0.0000 ft³/hr</td>
<td>200.2822 gpm</td>
</tr>
<tr>
<td>Density</td>
<td>0.0000 lb/ft³</td>
<td>8.3327 lb/gal</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Pressure taps</th>
<th>D and D/Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>Differential pressure</td>
<td>276.8000 in of water</td>
</tr>
<tr>
<td>Reynolds No.</td>
<td>137987.2230</td>
</tr>
<tr>
<td>Sm (Sizing parameter)</td>
<td>0.1309</td>
</tr>
<tr>
<td>Cd (Discharge coefficient)</td>
<td>0.6038</td>
</tr>
<tr>
<td>Beta ratio (d/D)</td>
<td>0.4603</td>
</tr>
<tr>
<td>Pipe inside diameter (D)</td>
<td>4.0260 in</td>
</tr>
<tr>
<td>Bore size (d)</td>
<td>1.8534 in</td>
</tr>
<tr>
<td>Kr (Flow resistance factor)</td>
<td>45.9733</td>
</tr>
</tbody>
</table>

**Figure 49 – Recommended orifice sizing**

The recommended bore diameter is 1.85 inches, and the flow resistance factor, Kr, is 46. Kr is used to insert the orifice into the pipe UnitOp that follows the pump.

Returning to the flowsheet, double-click pipe UnitOp 1, then select the Valves tab as shown in Figure 50. The value for Kr must be inserted; it is not automatically transferred from the orifice sizing procedure. Remember to fill in the Count box, as shown, to provide the number of orifices.

**Figure 50 – Adding Kr value to the Pipe specifications**

When you run the simulation, you will see that the frictional pressure drop across pipe UnitOp 1 has increased by 7.88 psi, while the Delta P across the valve has decreased by the same amount. The valve position has increased from 40.3% open to 43.6%, to adjust for the lower Delta P.
Part 2 of this paper will expand the scope of investigation to analysis of more complicated systems with branched piping.

Appendix: Referenced Tables

Table 2-2
Suggested Fluid Velocities in Pipe and Tubing

<table>
<thead>
<tr>
<th>Fluid</th>
<th>Suggested Trial Velocity</th>
<th>Pipe Material</th>
<th>Fluid</th>
<th>Suggested Trial Velocity</th>
<th>Pipe Material</th>
</tr>
</thead>
<tbody>
<tr>
<td>Acetylene (Observe</td>
<td>4000 fps</td>
<td>Steel</td>
<td>Sodium Hydroxide</td>
<td>6 fps</td>
<td>Steel</td>
</tr>
<tr>
<td>pressure limitations)</td>
<td></td>
<td></td>
<td>0—60 Percent</td>
<td>5 fps</td>
<td>and</td>
</tr>
<tr>
<td></td>
<td>4000 fps</td>
<td>Steel</td>
<td>50—75 Percent</td>
<td>4 fps</td>
<td>Nickel</td>
</tr>
<tr>
<td>Ammonia Liquid</td>
<td>6 fps</td>
<td>Steel</td>
<td>Sodium Chloride Sol’n.</td>
<td>5 fps</td>
<td>Steel</td>
</tr>
<tr>
<td></td>
<td>6000 fps</td>
<td>Steel</td>
<td>No Solids</td>
<td>(6 Min.—)</td>
<td>and</td>
</tr>
<tr>
<td>Ammonia Gas</td>
<td>6 fps</td>
<td>Glass</td>
<td>With Solids</td>
<td>15 Max.—</td>
<td>Nickel</td>
</tr>
<tr>
<td></td>
<td>2000 fps</td>
<td>Glass</td>
<td></td>
<td>7.5 fps</td>
<td>Steel</td>
</tr>
<tr>
<td>Chloroform Liquid</td>
<td>4 fps</td>
<td>Steel</td>
<td>Penclophon</td>
<td>6 fps</td>
<td>Moly or nickel</td>
</tr>
<tr>
<td>Chloroform Gas</td>
<td>6 fps</td>
<td>Copper &amp; Steel</td>
<td>Steam</td>
<td></td>
<td>Steel</td>
</tr>
<tr>
<td>Ethylene Chloride</td>
<td>4 fps</td>
<td>Glass</td>
<td>0—30 psi Sat.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ethylene Dichloride</td>
<td>6 fps</td>
<td>Steel</td>
<td>30—150 psi Sar.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ethylene Glycol</td>
<td>6 fps</td>
<td>Steel</td>
<td>150 psi up</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hydrogen Liquid</td>
<td>5 fps</td>
<td>Rubber Linde</td>
<td>Sulfuric Acid</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Gas</td>
<td>4000 fps</td>
<td>R. L. Saran</td>
<td>Sulfur Dioxide</td>
<td>4000 fps</td>
<td>Steel</td>
</tr>
<tr>
<td>Methyl Chloride Liquid</td>
<td>6 fps</td>
<td>Steel</td>
<td>Styrene</td>
<td>6 fps</td>
<td>Steel</td>
</tr>
<tr>
<td>Natural Gas</td>
<td>4000 fps</td>
<td>Steel</td>
<td>Trichloroethylene</td>
<td>6 fps</td>
<td>Steel</td>
</tr>
<tr>
<td>Oils, lubricating Oxygen</td>
<td>6 fps</td>
<td>Steel</td>
<td>Vinyl Chloride</td>
<td>6 fps</td>
<td>Steel</td>
</tr>
<tr>
<td>(Ambient temp.)</td>
<td>1800 fps Max.</td>
<td>Steel (300 ps Max.)</td>
<td>Vinylene Chloride</td>
<td></td>
<td></td>
</tr>
<tr>
<td>(Low temp.)</td>
<td>4000 fps</td>
<td>Type 304 SS</td>
<td>Water</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Propylene Glycol</td>
<td>5 fps</td>
<td>Steel</td>
<td>Average service</td>
<td>3–8 (avg. 6) fps</td>
<td>Steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Boiler feed</td>
<td>4–12 fps</td>
<td>Steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Pump suction lines</td>
<td>1–5 fps</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Sea and brackish</td>
<td>4eling</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Water, lined pipe</td>
<td>5–8 fps (3)</td>
<td>Steel</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Concrete</td>
<td>8–12 fps (Mn.)</td>
<td></td>
</tr>
</tbody>
</table>

Note: R. L. = Rubber-lined steel.

Table 2-2, Suggested Fluid Velocities in Pipe and Tubing (Ludwig, 1977)
## Table 2-7

**Flow of Water Through Schedule 40 Steel Pipe**

<table>
<thead>
<tr>
<th>Discharge</th>
<th>Pressure Drop per 100 feet and Velocity in Schedule 40 Pipe for Water at 60 F.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Gallons per Minute</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>1/16&quot;</td>
<td>0.000466</td>
</tr>
<tr>
<td>1/8&quot;</td>
<td>0.000866</td>
</tr>
<tr>
<td>1/4&quot;</td>
<td>0.001111</td>
</tr>
<tr>
<td>3/8&quot;</td>
<td>0.001207</td>
</tr>
<tr>
<td>1/2&quot;</td>
<td>0.001415</td>
</tr>
<tr>
<td>2&quot;</td>
<td>0.003137</td>
</tr>
<tr>
<td>3&quot;</td>
<td>0.005211</td>
</tr>
<tr>
<td>4&quot;</td>
<td>0.007491</td>
</tr>
<tr>
<td>5&quot;</td>
<td>0.009033</td>
</tr>
<tr>
<td>6&quot;</td>
<td>0.010485</td>
</tr>
<tr>
<td>8&quot;</td>
<td>0.013167</td>
</tr>
<tr>
<td>10&quot;</td>
<td>0.017030</td>
</tr>
<tr>
<td>12&quot;</td>
<td>0.021479</td>
</tr>
<tr>
<td>14&quot;</td>
<td>0.026308</td>
</tr>
<tr>
<td>16&quot;</td>
<td>0.031408</td>
</tr>
<tr>
<td>18&quot;</td>
<td>0.036798</td>
</tr>
<tr>
<td>20&quot;</td>
<td>0.042383</td>
</tr>
</tbody>
</table>

For pipe lengths other than 100 feet, the pressure drop is proportional to the length.
Thus, for 50 feet of pipe, the pressure drop is approximately one-half the value given in the table....for 300 feet, three times the given value, etc.

For pipe other than Schedule 40: v = $v_0$ (da/dp) and $\Delta P = a \Delta P_0$ (da/dp), where subscript $40$ refers to the conditions for schedule 40 pipe.

References


2. Technical Paper No.410, Crane Co., Engineering Division, Chicago, 1957


Author Information

Jeffrey L. Silverstein is a private Consulting Process Simulation Engineer & Analyst. He provides advanced process analysis, process simulation, and process design services to small and medium-sized chemical companies. He has 39 years of experience as a Senior Process Simulation Engineer with American Cyanamid Company and Cytec Industries, and served as the company expert on distillation, solvent recovery, and air emission control. In addition, he led CHEMCAD workshops to teach practical applications to plant engineers throughout the company. He holds a PhD in Chemical Engineering from The City University of New York. Dr. Silverstein can be reached by e-mail at JLS@ProcessSimulationExpress.com and by phone at 201-739-6850. He welcomes your inquiries regarding his services.
Using CHEMCAD for Piping Network Design and Analysis

Part 2 – Branched Piping Networks

By

Jeffrey L. Silverstein, PhD
Director
Process Simulation Simulation Express LLC
Ramsey, NJ 07446

Abstract

CHEMCAD is a powerful tool for designing and analyzing complex piping networks. It enables the engineer to integrate the piping network simulation into the overall process simulation, calculating a momentum balance in addition to a heat and material balance. This makes it possible to quickly evaluate the effects of changes in the process concept, or of its chemicals, on the piping network.

Part 1 of this paper presented a tutorial on how to use the piping network tool for unbranched pipe analysis. In Part 2 we will add complexity with more complicated networks, culminating in examples that demonstrate how CHEMCAD can be used to analyze practical plant piping problems.

Introduction

The CHEMCAD software suite is a powerful tool for designing and analyzing complex piping networks (1). In Part 1 of this paper, you saw how to use CHEMCAD to analyze unbranched flow problems. In this paper, Part 2, we will explore more complicated piping networks in which flow in a single pipe is divided between two or more downstream pipes, such as the distribution of cooling tower water to various plant heat exchangers and process vessels. We will also examine the combining of the flows from several pipelines into a single pipe, as might be seen in the return to a cooling tower. Indeed, complex piping systems are nothing more than compilations of sections of pipelines that involve dividing and combining flows.

We will begin by examining basic dividing and combining configurations, and see how these can be solved using CHEMCAD controllers. The two basic configurations will then be combined into a simple network. Again the solution will be demonstrated using CHEMCAD controllers.

Next we will explore the solution of piping network problems using a nodal method, and discuss simple rules for using nodes in CHEMCAD. Although CHEMCAD controllers can be used to solve piping networks, as the complexity increases it becomes more difficult to determine the position and settings for the controllers. In particular, specifying limits on flow rates that prevent flowsheet non-convergence from occurring can be challenging. Simulation of the piping network one section at a time to establish flow rate ranges for inclusion in the controllers of the flowsheet can be frustrating and time-consuming.

The simple rules for using nodes mitigate these issues, and provide for straightforward simulation of the entire flowsheet. However, with very complex networks, it is still advisable to build the network by converging it section by section.
Finally, we will look at practical plant piping examples, and see how CHEMCAD can be used to analyze these examples to understand and correct the plant issue.

**Simple Divided Flow Problem Using Controllers**

Let’s start with the simple divided flow problem with the filename #1 Simple Divided Flow - Controller Solution. The flowsheet is given in Figure 1.

![Simple Divided Flow - Controller Solution Flowsheet]

*Figure 1 – Simple divided flow problem flowsheet*

For details about the pump, heat exchanger, and piping specifications, see the simulation file. The temperature and pressure (80 °F and 14.7 psia, respectively) of the feed stream, stream 1, are fixed. The pressures of the exit streams are each fixed at 14.7 psia and the pressure drops of the heat exchangers are 3 and 2 psi, respectively, as noted in Figure 1. The outlet temperature of each exchanger is 100 °F.

The controllers are used to fix the pressure for each of the exit streams at 14.7 psia. Controller 7 adjusts the flow of stream 1 to accomplish this for stream 7. Controller 8 accomplishes this for Stream 8, by adjusting the flow ratio specification in the divider. A specified range of permissible divider settings is provided in the controller dialog box, namely, 0.10 – 0.90. The initial divider setting splits the flow equally to each branch.

Let’s set the flow for Stream 1 at 10 gpm, as shown in Figure 2.

![Specifications for stream 1]

*Figure 2 – Specifications for stream 1*
Running the simulation now results in the stream information given in Figure 3. The total flow is about 66 gpm, with 39 gpm going to the upper pipe and 27 gpm to the lower. This makes sense since the upper pipe is 100 feet shorter than the lower one and its elevation change is 5 feet lower.

**Simple Combining Flow Problem Using Controllers**

The filename for the simple combining flow problem is #2 Simple Combining Flow - Controller Solution. The flowsheet is given in Figure 4. For details about the piping and other UnitOps, see the simulation file.

**Figure 3 – Flow summary, simple divided flow controller solution**

**Figure 4 – Flowsheet for simple combining flow using controllers**

As noted on the flowsheet, there are seven independent variables and four independent equations that define the flow problem. Three independent variables must be specified to permit solution. These are the pressures of the feed streams and the exit stream.

Controller 1 adjusts the flow of stream 2 to fix the outlet pressure at 20 psig. Controller 3 adjusts the flow of Stream 1 so that the pressures of streams 5 and 6 are equal as they enter the mixer, UnitOp 8. The initial setting for the pressure outlet of the mixer is left blank in the mixer dialog box.

Set the flow for streams 1 and 2 at 10 gpm, as shown in Figure 5.
Running the simulation now results in the stream information given in Figure 6. The total flow is about 13.3 gpm, with 7.7 gpm flowing through the upper pipe and 5.7 gpm through the lower. This makes sense, since the lower pipe has a larger equivalent length.

**Figure 5 – Feed stream specifications**

<table>
<thead>
<tr>
<th>Stream No.</th>
<th>1</th>
<th>2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stream Name</td>
<td>F1?</td>
<td>F2?</td>
</tr>
<tr>
<td>Temp °C</td>
<td>25.0024</td>
<td>25.0024</td>
</tr>
<tr>
<td>Pres psig</td>
<td>50.0000</td>
<td>50.0000</td>
</tr>
<tr>
<td>Enth kBtu/h</td>
<td>-19056.</td>
<td>-25795.</td>
</tr>
<tr>
<td>Vapor mole frac.</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>Total lbmol/l</td>
<td>155.1468</td>
<td>210.0087</td>
</tr>
<tr>
<td>Total lb/l</td>
<td>2794.9690</td>
<td>3783.3074</td>
</tr>
<tr>
<td>Total std L/gpm</td>
<td>5.5519</td>
<td>7.5557</td>
</tr>
<tr>
<td>Total std V/scf</td>
<td>53702.08</td>
<td>75399.90</td>
</tr>
<tr>
<td>Flow rates in lb/h</td>
<td>2794.9690</td>
<td>3783.3074</td>
</tr>
<tr>
<td>Water</td>
<td>2794.9690</td>
<td>3783.3074</td>
</tr>
</tbody>
</table>

**Figure 6 – Flow summary for simple combining flow controller solution**

**Simple Network Using Controllers**

We now combine the simple divided flow flowsheet with the simple combining flow flowsheet to construct a simple network, as shown in Figure 7. The filename is #3 Simple Network – Controller Solution. For details about the piping and other UnitOps, see the simulation file.
This flowsheet combines the two previous ones, to construct a simple network similar to a cooling system network in a plant. A cold water stream at 80 °F from the cooling tower is split to provide cooling for two heat exchangers. The streams are recombined and returned to the cooling tower.

We need to know the flow rate provided by the pump, the pump outlet pressure, the flow rates of the two streams, and the pressure after they recombine. There are only two independent variables: one at the system inlet and one at the system outlet. When these are specified, all of the flow-related variables between them, the “interior variables,” can be calculated.

In this example, the independent variables are the system inlet and outlet pressures. Controller 13 iteratively calculates the stream split until the respective pressures of the streams are equal when they recombine. Controller 14 iteratively calculates the flow rate into the pump until the system outlet pressure equals 14.7 psia, the value fixed initially as one of the independent variables.

If either of the independent variables is changed, all of the “interior variables” will also change. It is not surprising, however, that if the system inlet and outlet pressures are equal, the calculated flow results are independent of the actual value of these pressures. This is shown in Figure 8, for inlet and outlet pressures of 14.7 psia and 64.7 psia, respectively.

**Figure 8 – Flow summaries for simple network using a controller**

**Figure 9** provides the pump characteristics for each case. Because both pressures were increased by the same amount, the resistance to flow that the pump experiences does not change and it operates at the same point on the pump curve. The NPSHA, however does change, because the upstream pressure has increased.
Figure 9 – Comparison of pump characteristics for the two cases

Solving Piping Problems with Controllers - Summary

We have demonstrated with three simple examples that controllers can be used to solve increasingly complicated piping problems. Furthermore, we have shown that the number of independent variables that must be specified to solve such problems is equal to the total number of independent system inlet and outlet streams.

For the simple divided flow problem and the simple combining flow problem, there are three independent variables, as shown in Figure 1 and Figure 4, respectively. In each case we specified the system inlet and outlet pressures. The simple network problem of Figure 7 has only one inlet and one outlet stream, and only two independent variables.

Using CHEMCAD, it is possible to add any number of simple network piping configurations together to develop and simulate complex piping systems used in plant-wide cooling tower networks, raw material distribution networks, solvent collection systems, liquid waste collection, and vapor emissions to flare or scrubber systems.

In each case, the number of degrees of freedom is determined by the number of independent inlet and outlet streams. As the systems become more and more complex, however, the difficulty in simulation lies in determining the location and settings of the controllers internal to the system that are required to complete the calculation. This can be quite frustrating and time-consuming.

Using Nodes for Piping Network Simulation

CHEMCAD offers a simple and robust alternative to using controllers that does not require the details of feedback controller calculations to be consciously considered. This approach uses nodes. It is only necessary to determine and set the values of the system independent variables at each of the independent inlet and outlet streams of the system. As will be shown, although nodes are required to be placed at “interior” positions in the piping network flowsheet, their settings are all identical and invariant.

The use of nodes for piping networks originated with Hardy Cross, a structural engineering professor at the University of Illinois at Urbana–Champaign. The Hardy Cross Method, first published in 1936, is an iterative method for determining the flow in pipe network systems where the inputs and outputs are known, but the flow inside the network is unknown (2). Other investigators (3 - 6), have sought to improve on the original method over the years.
Let’s begin with a single pipe element, as shown in Figure 10. The corresponding filename is #4 Single Pipe With Nodes.

The NODE UnitOp

1. Allows you to specify the pressure on either side of a UnitOP and calculate the flow rate as a function of pressure.
2. As an option, you may specify one pressure and the flow rate
3. A series of UnitOps may be connected using several nodes.

Three Variables: F1, P1, P2
One Equation: \( F_1 = f(P_1, P_2) \)

Must specify 2 variables to constrain the system: \( F = 20 \text{ gpm water at } 35\text{C}; P_2 = 14.7 \text{ psia} \)

What is \( P_1 \)?

Piping Run: 1 horizontal run of 200 ft, 2-in Sch 40 pipe with 2 - 90 degree elbows
1 vertical run of 200 ft with one ball valve and 3 - 90 degree elbows

Figure 10 – Single pipe with nodes flowsheet

This is a simple example of nodes to demonstrate their use. There are three variables that define the flow problem: the flow rate and the inlet and outlet pressures. There are two degrees of freedom for this problem. We must fix these degrees of freedom by specifying any two of the three variables. The third variable can then be calculated.

The dialog box for the node in the feed stream is shown in Figure 11.

Figure 11 – Node dialog box for inlet with fixed flow rate
There are two input choices for this inlet node. You can specify either the system inlet pressure or the inlet flow rate. For this example, fix the inlet flow rate at 20 gpm. Mass and molar flow units can also be specified, if desired. The pressure specification is *Variable pressure* because this value will be calculated.

Alternatively, you can fix the pressure and fill in a value for it, and let the flow rate be the calculated variable by designating the stream into the node as a *Free inlet stream*. This is shown in Figure 12.

![Figure 12 – Node dialog box for inlet with fixed pressure](image)

In both cases, indeed for every inlet node, the outlet stream flow—the downstream pipe in this case—is set to *Flow set by UnitOP*.

Now let’s look at the settings for the outlet node, shown in Figure 13.

![Figure 13 – Node dialog box for outlet with fixed pressure](image)
There are two input choices for this outlet node. You can specify either the system outlet pressure or the outlet flow rate. For this example, fix the outlet pressure at 14.7 psia. The outlet flow specification is set to Free outlet stream, as the flow rate has already been specified in the inlet node. For every outlet node, the inlet stream flow specification is Flow set by UnitOP, the upstream pipe in this case.

It is important to note that the piping elevation at the physical location of each node in a flowsheet needs to be specified in the Node dialog box, as it is in Figure 13, for the outlet node. The pipe dialog box shows an elevation of 200 feet that must be indicated for the node following the pipe. For a series of pipes with changing elevations, the elevation at a given node is the sum of the elevations of the pipes upstream of it.

Let’s look at the Convergence dialog box, as shown in Figure 14, accessed from the main CHEMCAD menu by selecting Run > Convergence. Note that the calculation sequence is Simultaneous modular. When a piping simulation is set up in CHEMCAD with nodes, the calculation sequence automatically defaults to this setting.

![Figure 14 – The Convergence Parameters dialog box with Simultaneous modular calculation sequence selected](image)

If the pipe dialog box is open, as shown in Figure 15, we see that the sizing option used is Rating (default). CHEMCAD suggests that for piping simulations with nodes, the sizing option choice should be: Given size, Pin and Pout, calc flow rate. In my personal experience, I have found that both of these sizing options work. However, since Chemstations recommends the latter, I suggest that the reader follow it.
Let’s now run the 20 gpm flow rate case. The required upstream pressure is 106.7 psia. Open the pipe dialog box to check results, as shown in Figure 16.

Figure 15 – Pipe dialog with Rating (default) sizing option selected

Figure 16 – Calculated results for the pipe UnitOp

To have flow through this pipe, enough upstream pressure needs to be applied to overcome the 200 feet of elevation, corresponding to 86.6 psia. What happens if the upstream pressure is 75 psia and the downstream pressure is 15 psia? The Delta P at 60 psia is not sufficient to overcome the piping elevation change, and no flow would occur. Water would enter the pipe and rise to height corresponding to the 60 psia pressure differential, roughly 138 feet. How does CHEMCAD handle this situation?

Let’s set the inlet pressure at the Inlet Node to 75 psia, as shown in Figure 17, and change the outlet pressure at the Outlet Node to 15 psia.
When this simulation runs, it converges mathematically. However, CHEMCAD calculates a negative flow rate. The calculated inlet and outlet streams are given in Figure 18.

**FLOW SUMMARIES:**

<table>
<thead>
<tr>
<th>Stream No.</th>
<th>2</th>
<th>3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stream Name</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Temp °F</td>
<td>59.9946</td>
<td>59.9947</td>
</tr>
<tr>
<td>Pres psia</td>
<td>75.0000</td>
<td>15.0000</td>
</tr>
<tr>
<td>Enth kBTu/h</td>
<td>1.6001E+05</td>
<td>1.6001E+05</td>
</tr>
<tr>
<td>Vapor mol fr.</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>Total lbmol/h</td>
<td>-1299.4570</td>
<td>-1299.4570</td>
</tr>
<tr>
<td>Total lb/h</td>
<td>-23409.7168</td>
<td>-23409.7168</td>
</tr>
<tr>
<td>Total std L gpm</td>
<td>-46.7518</td>
<td>-46.7518</td>
</tr>
<tr>
<td>Total std scfh</td>
<td>-493116.16</td>
<td>-493116.16</td>
</tr>
<tr>
<td>Flow rates in lb/h</td>
<td>-23409.7168</td>
<td>-23409.7168</td>
</tr>
<tr>
<td>Water</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Figure 18 – Flow summaries for pipe inlet and outlet streams**

**Rules for Using Nodes**

Now let’s define some simple rules and strategies for using nodes in complex piping networks. There are three combinations of settings that are typical for inlet and outlet nodes, as shown in Figure 19. In each case two variables are fixed and one is calculated. Figure 20 provides some guidelines for complex piping network simulations with CHEMCAD.
Some Rules For Simulating Piping Networks With Nodes

1. Option 5 should be chosen for pipe simulators: "Given size, Pin and Pout, calc flow rate"

2. Interior nodes are always specified as "Flow set by UnitOP" for the inlet and outlet streams and the pressure is always Variable.

3. The degrees of freedom for the problem are set by constraints at the process Inlet and Outlet Nodes

4. The RUN mode for the simulation must be Simultaneous Modular

Settings For Inlet And Outlet Nodes

![Diagram of settings for inlet and outlet nodes]

Some Guidelines for Piping Network Simulations With Chemcad

1. Develop a general diagram of the system, designating blocks or areas of the flowsheet which will be developed in Chemcad one at a time.

2. Use an easy to follow coding system to relate the actual piping elements in the plant with their representation in Chemcad.

3. For complex systems, build the Chemcad flowsheet in small segments, converging each increasingly more complicated flowsheet before adding additional piping sections. This allows easier troubleshooting for errors.

4. When possible, provide reasonable guesses for pressures at internal nodes, based on converged calculations made previously for adjacent nodes. This might help make convergence easier.

5. Be sure that the elevation for nodes are correct and are compatible with that given for elevation changes in adjacent piping.

6. If the calculation seems to be proceeding but convergence does not occur, try simply to rerun it.

7. If a singular matrix error message occurs:
   a) print the stream summary and check for anomalies, like negative flow rates. The pressure drop may be too high for units for which it is specified and not calculated, like heat exchangers.
   b) try rerunning the simulation
   c) try relaxing the convergence specification
   d) provide reasonable pressure estimates for internal nodes in areas of the flowsheet which have not already converged

8. Save all converged "simpler" flowsheets as you build up to more complicated ones. They provide a converged starting point for the next more complex flowsheet which may be hopelessly unconverted. Retreat to a safe position and start again.

9. For internal nodes the input is ALWAYS: Variable Pressure, Flow Set by Unit Op, In and Out

10. Constraints on the system are ALWAYS set by the external nodes

![Diagram of guidelines for building piping network simulations using CHEMCAD]
Simple Network Using Nodes

Now let’s go back to the simple network problem and solve it with nodes. The flowsheet is given in Figure 21. The filename is #5 Simple Network - Solve With Nodes.

![Simple Network Using Nodes Flowsheet](image)

**Figure 21 – Solving the simple network problem using nodes**

There is one inlet stream and one outlet stream, so there are two degrees of freedom. The system inlet and outlet pressures are specified, as shown in the inlet and outlet node dialog boxes, given in Figure 22 and Figure 23, respectively.

![Inlet Node Dialog Box](image)

**Figure 22 – Specifying system inlet pressure**

![Outlet Node Dialog Box](image)

**Figure 23 – Specifying system outlet pressure**
Figure 23 – Specifying system outlet pressure

The system inlet and outlet flows are designated as free streams, to be calculated by CHEMCAD. All of the internal nodes have identical settings: variable pressure and flow in and flow out set by the upstream and downstream UnitOp, respectively. In flowsheets that are difficult to converge, it is often helpful to estimate the pressure at internal nodes, as indicated in the guidelines provided in Figure 20. Note that the elevation values shown in the Node dialog boxes reflect the elevation changes of the pipe sections in the flowsheet.

Let’s set the flow rate for stream 1 at 10 gpm, as shown in Figure 24, and run the simulation. Figure 25 shows a comparison of the results for the solution using controllers and that using nodes. They are identical, as one would expect.

Figure 24 – Setting stream 1 flow rate to 10 gpm
Figure 25 – Comparison of using controllers and nodes, showing that results are identical

Now, let’s end this paper with two examples that demonstrate how CHEMCAD’s piping package can be used to analyze real plant problems.

Lower Than Expected Raw Material Delivery Rate

An acrylic acid (AA) off-loading piping system has been designed for a plant located in Europe. The design flow rate is 50,000 L/hr (220 gpm) so that the delivery truck could be off-loaded in no more than 30 minutes. The actual observed rate after start-up was about 35,000 L/hr, resulting in an off-load time of about 43 minutes. Why is the flow rate lower than expected, and how can it be improved without incurring a large cost?

The AA off-loading piping system, as designed, is shown in Figure 26.

Figure 26 – Flowsheet of acrylic acid off-loading piping system

Acrylic acid is delivered to the plant by truck. The operator uses flexible hose to connect to the hard pipe at the inlet to the plant acrylic acid pump. The flow out of the pump has two branches; the first goes to the plant’s acrylic acid storage tank, and the second goes to the acrylate process.
When the storage tank is being loaded, the flow to the process is blocked. The same pump is used to feed the process from the storage tank. So, when acrylic acid is delivered, it takes two paths to the storage tank. The CHEMCAD file for this design case is **#6 AA Off-load System – Design Case**. The details of the piping can be reviewed by opening the dialog boxes for each of the UnitOps.

The inlet and outlet nodes, respectively, set the inlet and outlet pressures at 14.7 psia. The 3” control valve has a Cv of 75 and is set at 100% open, as shown in Figure 27.

![Figure 27 – Control Valve dialog box for AA off-load design case](image)

When you run the flowsheet, you see that the maximum flow to the AA storage tank is 57,128 L/hr, which will deliver the desired quantity of AA in less than 30 minutes. The results are given in Figure 28.

![Figure 28 – Maximum flow to AA tank](image)

The piping design as conceived is adequate for delivering the desired flow. What could be restricting the AA flow?

One possibility is that the installed valve trim is not right. Suppose a restricted capacity trim was supplied. Many restricted capacity trim combinations are designed to furnish about 40% of full-size trim capacity. Let’s suppose this occurred in this plant, and run another simulation with the valve position set at 40% open. The filename is **#7 AA Off-load System – Reduced Valve Trim**, and the Control Valve dialog box is shown in Figure 29.
When you run this flowsheet, you see that the flow to the AA storage tank is now 35,458 L/hr. The results are given in Figure 30.

**FLOW SUMMARIES:**

<table>
<thead>
<tr>
<th>Stream No.</th>
<th>Stream Name</th>
<th>To AA Tank</th>
<th>Temp (°F)</th>
<th>Porsa (psia)</th>
<th>Enth (MMBtu/h)</th>
<th>Vapor mol frac.</th>
<th>Total kmol/h</th>
<th>Total kg/h</th>
</tr>
</thead>
<tbody>
<tr>
<td>19</td>
<td></td>
<td></td>
<td>77.2488</td>
<td>14.7000</td>
<td>-190.65</td>
<td>0.00000</td>
<td>520.9463</td>
<td>37541.4766</td>
</tr>
<tr>
<td>Total std L liter/</td>
<td>35458.4375</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Total std V scFt</td>
<td>435827.41</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flow rates in kg/h</td>
<td>37541.4766</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Acrylic Acid**

This flow is consistent with what the plant actually experienced. When the valve manufacturer was called in, it was found that, indeed, a restricted capacity trim was supplied. When the valve was replaced with one having the correct trim, the desired flow was achieved.
Can a feed line to a scrubber be tied in to an existing process feed line?

An existing line feeds molar excess methanol from the recovered methanol storage tank to a process that uses methanol as a reactant. The piping arrangement is shown in Figure 31. Details can be found in the CHEMCAD file called #8 Original Recovered Methanol Line To Process.

Heretofore, the plant has used fresh methanol as the scrubbing fluid for vapor vented to the atmosphere. It has been determined that it is economically advantageous to use recovered methanol instead of fresh as the scrubbing fluid.

The plant would like to tie in a new feed line to the scrubber from the recovered methanol line. The plant suggested tie-in is depicted in Figure 32. The filename is #9 Proposed Plant Piping Tie-In.

Before running this simulation, first isolate the piping tie-in and determine the supply pressure needed to feed 3 gpm of methanol to the scrubber. The piping layout is shown in Figure 33, and the filename is #10 Methanol Scrubber Feed Line – Isolated.
When you run this simulation with a fully open control valve, you will see that the minimum supply pressure at node 19 is 11.9 psig. Most of the pressure is required to overcome the elevation increase of the piping.

When you run the simulation called #8 Original Recovered Methanol Line To Process, as you can see from Figure 31, the pressure at the proposed tie-in point is only 9.13 psig. Consequently, the chosen tie-in point will not work. If you now run the simulation called #9 Proposed Plant Piping Tie-In, you will see that the simulation does not converge; the results show negative pressure in the new line. The required pipe line outlet pressure, fixed at 0.2166 psig, cannot be achieved.

Two solutions were offered. As shown in Figure 34, the first relocates the new piping to the scrubber to a tie-in point upstream of control valve FIC-1. This creates a parallel flow line and requires additional piping as shown in the figure. The filename is #11 Relocate Proposed Plant Piping Tie-In.

In this case, there is more than enough pressure at node 34 to overcome the change in piping elevation and supply 3 gpm of methanol to the scrubber. Note that most of the pressure drop in the line is taken across the control valve.

The second approach is to add a booster pump to the scrubber feed line. This is shown in Figure 35. The filename is #12 Proposed Plant Piping Tie-In Add Pump. The booster pump is added to raise the supply.
pressure to 20 psig. The plant chose this option, rather than the additional piping, since the cost was lower.

Figure 35 – Flowsheet showing addition of a booster pump to the scrubber feed line

In both this example and the previous one, CHEMCAD provided a powerful tool for analyzing the existing piping layouts and for simulating alternatives for corrective action. In each case, the analysis led to a successful modification of the piping system so that plant objectives could be met.

Summary

Part 1 of this paper explained how the various CHEMCAD UnitOps that relate to piping analysis can be configured to simulate an unbranched piping system. It started with a detailed look at the Pipe UnitOP and various methods for pipe sizing available in CHEMCAD. Results were compared to standard tabulated results for water with excellent agreement.

A pump UnitOP was added to the pipe to determine what flow would be achieved with the corresponding pump curve. The meaning of NPSHA was discussed.

A control valve UnitOP was inserted into the simulation to demonstrate how it could be used to achieve a desired flow rate. Important considerations for valve selection were discussed. It was shown how a restriction orifice can be added to the pipe line. The effect on valve position to maintain the desired flow rate was demonstrated.

Part 2 extended the discussion to branched piping systems. It began with discussions on the use of controllers for solving simple divided flow and simple combining flow problems. These piping configurations were then combined into a simple network. Again it was shown how controllers can be used to simulate this system.

As the complexity of such systems increases, as in a cooling tower piping network, it becomes increasingly difficult to properly locate the controllers and determine the correct settings for an easy solution. To overcome these difficulties, the use of a nodal solution approach was introduced. Nodal positioning and settings were discussed for a single pipe. Guidelines for the use of nodes in complex networks were provided. The simple network problem previously solved with controllers was solved using nodes. The solutions were compared and found to be identical.

Two actual plant piping problems were then presented. It was shown how CHEMCAD was used to analyze them and to determine design modifications that resulted in meeting plant objectives.
References


Author Information

Jeffrey L. Silverstein is a private Consulting Process Simulation Engineer & Analyst. He provides advanced process analysis, process simulation, and process design services to small and medium-sized chemical companies. He has 39 years of experience as a Senior Process Simulation Engineer with American Cyanamid Company and Cytec Industries, and served as the company expert on distillation, solvent recovery, and air emission control. In addition, he led CHEMCAD workshops to teach practical applications to plant engineers throughout the company. He holds a PhD in Chemical Engineering from The City University of New York. Dr. Silverstein can be reached by e-mail at JLS@ProcessSimulationExpress.com and by phone at 201-739-6850. He welcomes your inquiries regarding his services.