

FluidFlow

The Flow of Gas via Pressure Difference

© Flite Software NI Ltd



FluidFlow

The Flow of Gas via Pressure Difference

Problem

Calculate the stagnation pressure loss, friction loss, flow rate, velocity, Reynolds number, and friction factor for a 20 m long 4" schedule 40 steel pipe transporting air at 25°C. The pipe inlet pressure shall be 1.1 barg with an outlet pressure of 1 barg.

Objective

The fluid will flow naturally from the source with higher pressure to the sink with lower pressure. The difference in pressure is called pressure head.

In this example, you will build a model to illustrate the flow of gas via pressure difference.

This example includes the following tasks:

- Task 1 – Add the Boundaries and Piping Component
- Task 2 – Define the Boundaries, Fluid, and Piping
- Task 3 – Select Results Unit Set
- Task 4 – Calculate and Perform Hydraulic Analysis
- Task 5 – Compare the FluidFlow Results to Hand Calculation

Task 1 – Add the Boundaries and Piping Component

1. Start a new flowsheet by opening the FluidFlow software. The user interface will appear as shown below (Figure 1).

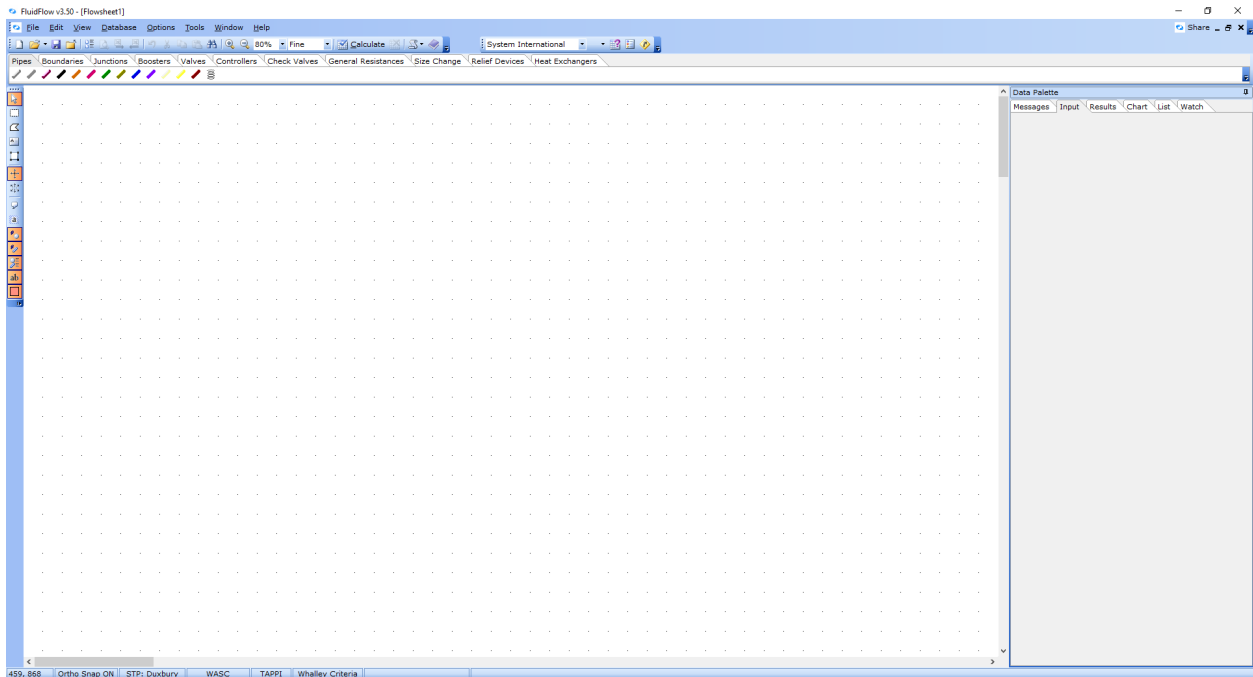


Figure 1: New flowsheet user interface.

2. Consider the boundary nodes to use for the system. Since we know the inlet and outlet pressure, we can use the *Known or Assigned Pressure* node available from the *Boundaries* tab on the *Component Palette* (Figure 2).

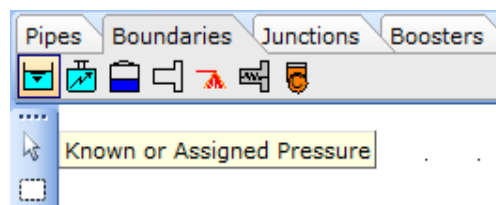


Figure 2: Known or Assigned Pressure Node.

3. Place two of these nodes (inlet & outlet) on the flowsheet by left mouse-clicking on the icon on the *Component Palette*. To place the node on the flowsheet we need to left mouse-click in the desired location (Figure 3).



Figure 3: Known Pressure Nodes.

4. This represents our pipe boundaries. We can now connect the boundaries by selecting the desired pipe material from the *Component Palette*. In this design case, we know the pipe is a Schedule 40 steel pipe. Therefore, we need to select the Steel Pipe or Duct icon from the Component Palette (Figure 4).

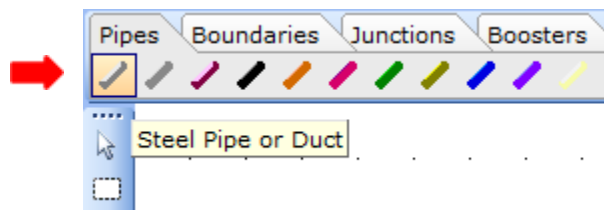


Figure 4: Steel Pipe or Duct Icon.

5. To connect the boundaries, left mouse-click directly over the inlet boundary node (node 1) and again, left mouse-click directly over the outlet boundary node (node 2). Note, when you position the mouse selector above the boundary node on the flowsheet, you will see the selector change to a green “tick” symbol (✓). This indicates that the software is ready to connect our new node.

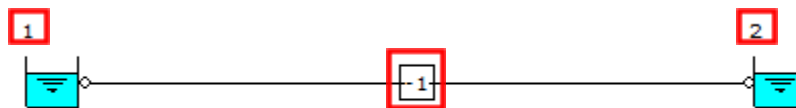


Figure 5: Hydraulic Model.

6. At this stage, we will notice that the software automatically assigns a unique *User Number* (reference Number) to each node placed on the flowsheet. All boundaries and fittings have positive *User Numbers* whereas all pipes have negative *User Numbers*. In Figure 5, we can see that the inlet and outlet boundaries are assigned with *User Number* “1” & “2” respectively and the single pipe is assigned with *User Number* “-1”. Note, if these reference values have not appeared automatically, they may be toggled off. You can toggle these back on by selecting the buttons to the left of the flowsheet titled “Show or Hide Node Numbers” and “Show or Hide Pipe Numbers”. The buttons appear as follows (Figure 6):

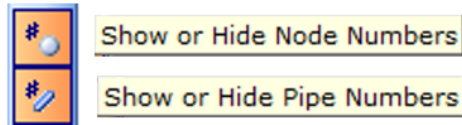


Figure 6: Show or Hide Node/Pipe Numbers buttons.

The automatic assignment of *User Numbers* helps us differentiate each of the nodes when we examine our systems – post calculation. This will become more apparent later.

7. Save your flowsheet as **04-FF Gas via Pressure Difference.FF3**

Task 2 – Define the Boundaries, Fluid, and Piping

The next step is to edit the default data for each of the nodes placed on the flowsheet (Boundaries, Fluid, and Piping). Let us begin with the boundaries. We know the intended pressure units are in **barg** for both the inlet and outlet boundary. We can edit multiple nodes at once which will help speed up model development.

8. Left mouse-click on the inlet boundary (node 1) and whilst holding the SHIFT key, left mouse-click on the outlet boundary (node 2). You should now see both nodes highlighted on the flowsheet (Figure 7).

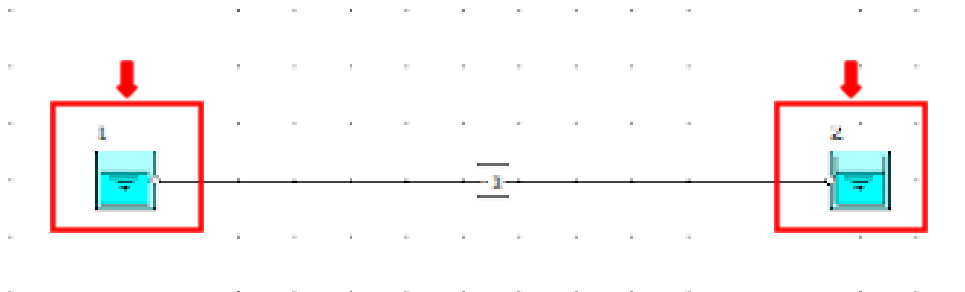


Figure 7: Highlighted Nodes.

9. We can now edit the data entry for both nodes at once from the Input Inspector on the *Data Palette* (Figure 8).

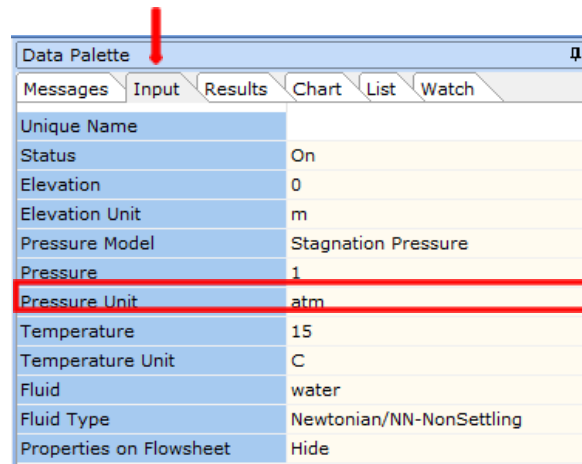


Figure 8: Input Inspector.

10. In this case we need to change the *Pressure Unit* from the default **atm** setting to **barg**. Left mouse-click on the *Pressure Unit* field and you will see a drop-down arrow symbol appear on the right-hand side. Click on this symbol and a drop-down menu will appear showing the various units you can choose from. Select **barg** from the list (Figure 9). We have now successfully changed the pressure units for both nodes.

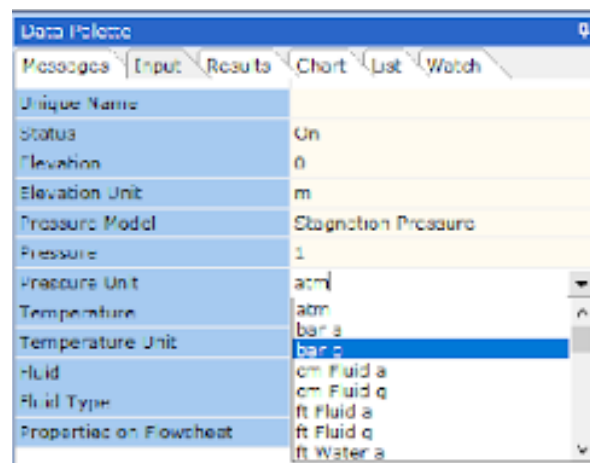


Figure 9: Pressure Units.

11. We can now change the inlet operating pressure, temperature, and fluid by selecting node 1. View the *Data Palette* and click the *Input Tab*. Enter the following specifications:

In this cell...	Enter...
Pressure	1.1
Temperature	25
Fluid	air

To change the default fluid water, left click on the field titled *Fluid* and a button with 3 dots will appear (Figure 10):

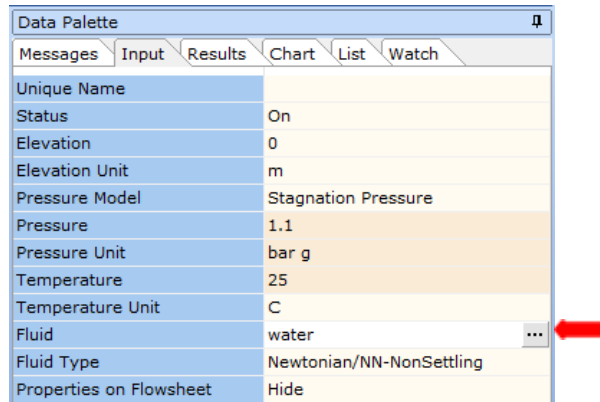


Figure 10: Fluid with 3 dots button.

Click the button and a *Select Fluid* dialog box will appear (Figure 11):

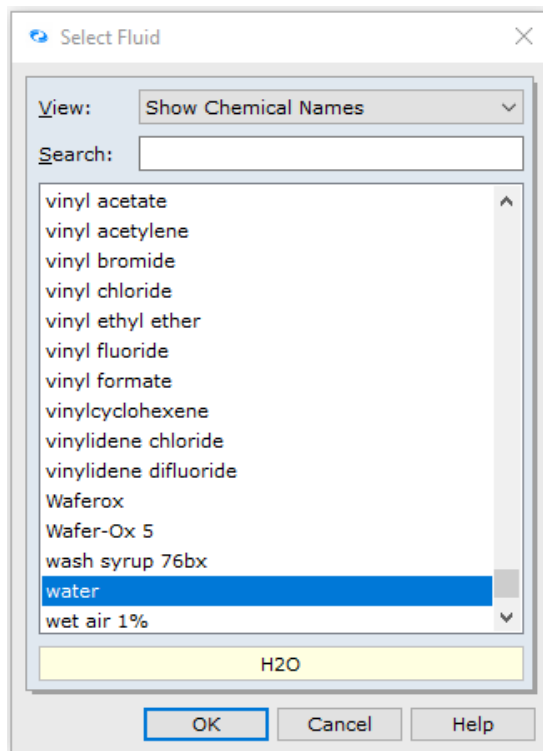


Figure 11: Select Fluid Dialog Box.

This dialog box will allow us to access the fluid database (Figure 11). From the list of the available fluids, you may browse via scroll bar or search for the **air** via the search field. Select **air** and click **OK** to complete this change (Figure 12).

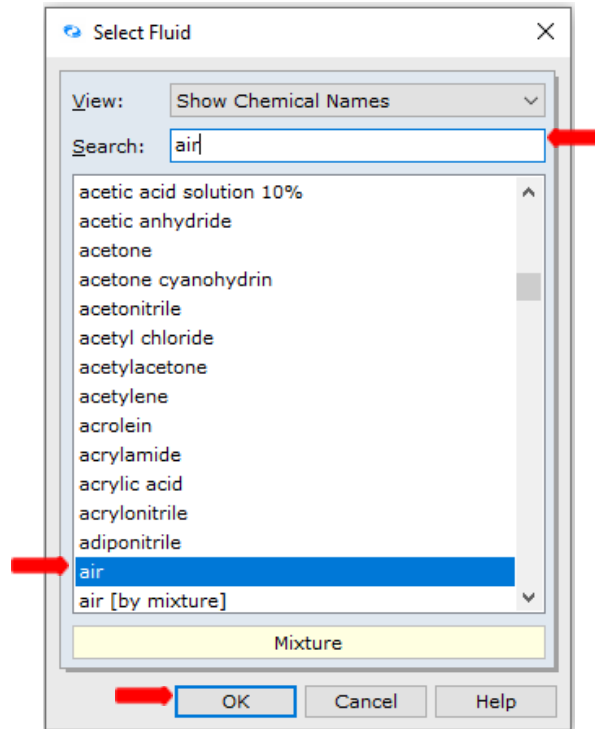


Figure 12: Select Fluid Dialog Box.

The Input Inspector on the *Data Palette* should now look like the screenshot given below (Figure 13).

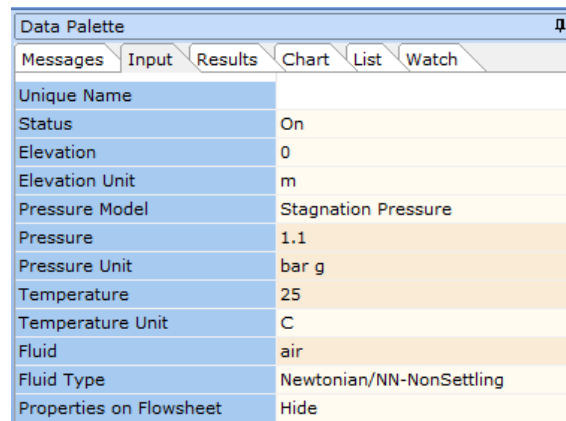


Figure 13: Input Inspector at Node 1.

12. Let us now check the input values in node 2 (Figure 14).

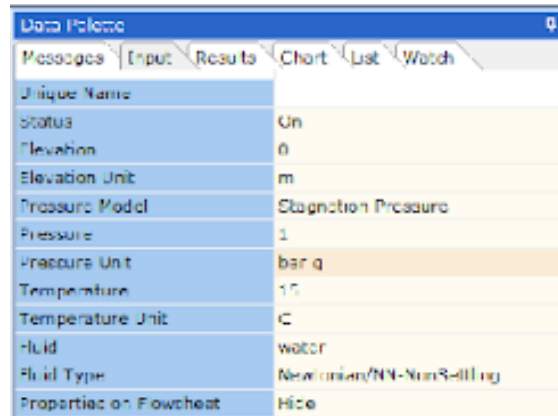


Figure 14: Input Inspector at Node 2.

The operating pressure is already at 1 barg which is consistent with our specification for node 2. For the temperature and fluid inputs, we don't need to change it to 25°C and air, respectively. Note that FluidFlow will use the operating conditions of the higher pressure node 1 in calculating the fluid properties such as density, viscosity, thermal conductivity, etc.

- Let us now edit the default pipe data by selecting pipe number -1. You should now see that pipe number -1 is highlighted on the flowsheet (Figure 15).

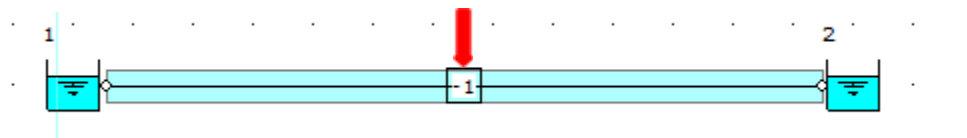


Figure 15: Highlighted Pipe.

- View the *Data Palette* and click the *Input Tab*. Enter the following specifications:

In this cell...	Enter...
Length	20
Nominal Size	4 inch

To change the default 2-inch size, left click on the field titled *Nominal Size* and a button with 3 dots will appear (Figure 16):

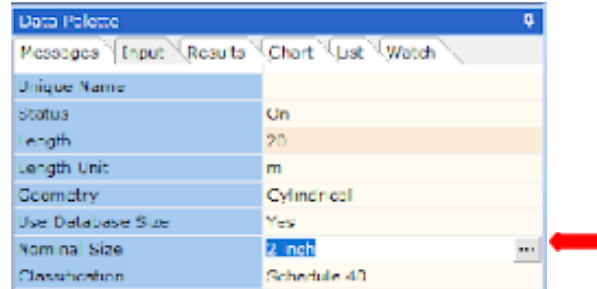


Figure 16: Nominal Size with 3 dots button.

Click the button and a *Set Pipe Dimension* dialog box will appear (Figure 17):

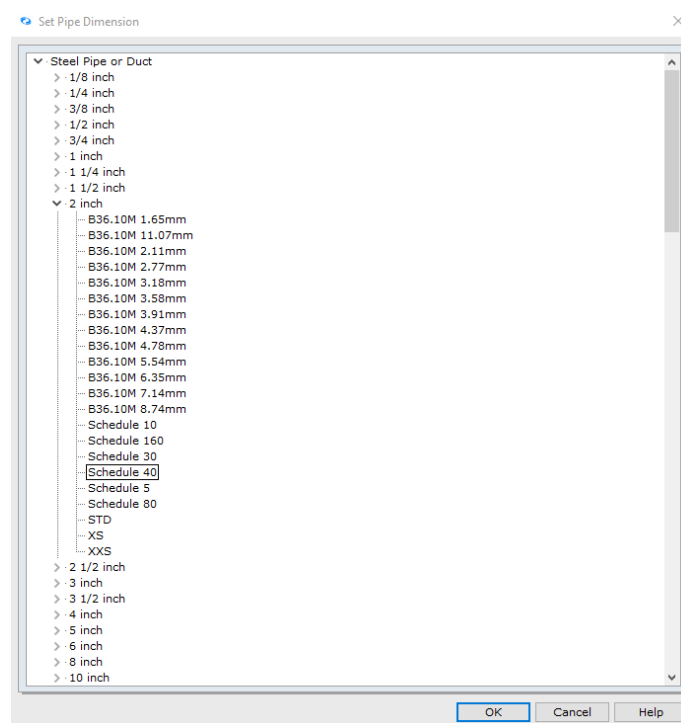


Figure 17: Set Pipe Dimension Dialog Box.

This dialog box will allow us to access the pipe database (Figure 17). From the list of the available pipe sizes, click on the **4 inch** to view the various pipe classifications for this diameter. Select **Schedule 40** and click **OK** to complete this change (Figure 18).

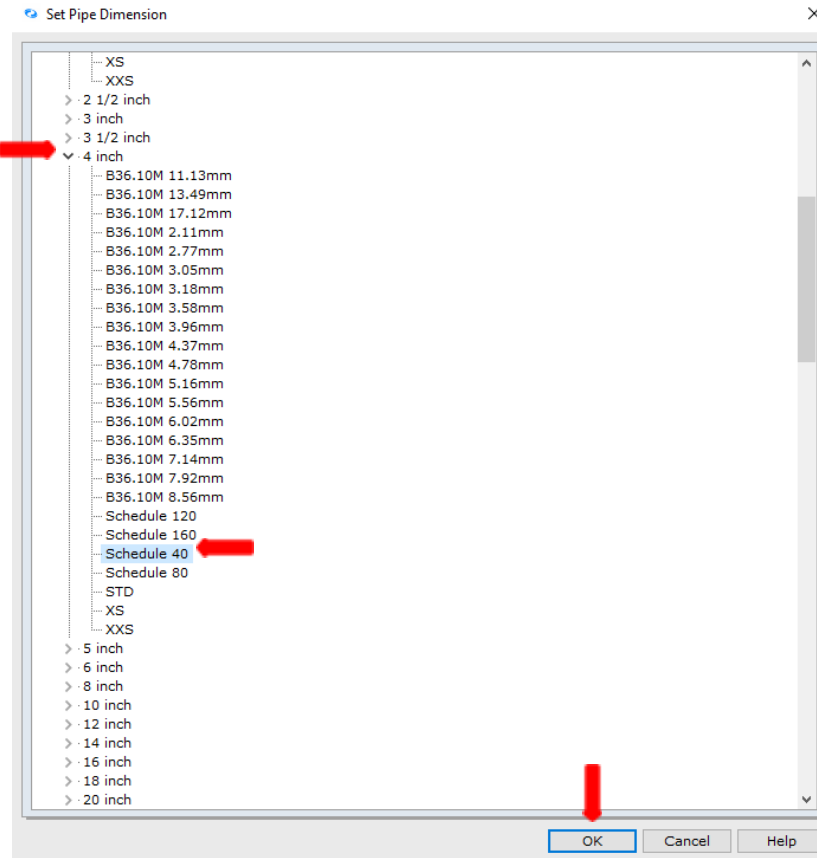


Figure 18: Set Pipe Dimension Dialog Box.

15. The Input Inspector on the *Data Palette* should now look like the screenshot given below (Figure 19).

Data Palette	
Messages Input Results Chart List Watch	
Unique Name	
Status	On
Length	20
Length Unit	m
Geometry	Cylindrical
Use Database Size	Yes
Nominal Size	4 inch
Classification	Schedule 40
Friction Model	Moody
Use Database Roughness	Yes
Roughness Description	Clean or new
Use Database Scaling	No
Scaling (0 to 50%)	0
Sizing Model	Economic Velocity
Heat Loss Model	Ignore Heat Loss/Gain
Draw Thickness [1..5]	1
Draw Color	cBlack
Properties on Flowsheet	Hide

Figure 19: Input Inspector at Pipe Number "-1".

Note: The entire system is horizontal and as such, we have left all node elevations as 0 m.

16. **Save** your flowsheet.

Task 3 – Select Results Unit Set

In FluidFlow, it is possible to change the default results unit set used to display variables.

17. View the top portion of the user interface. The Units selector is on the middle portion (Figure 20):

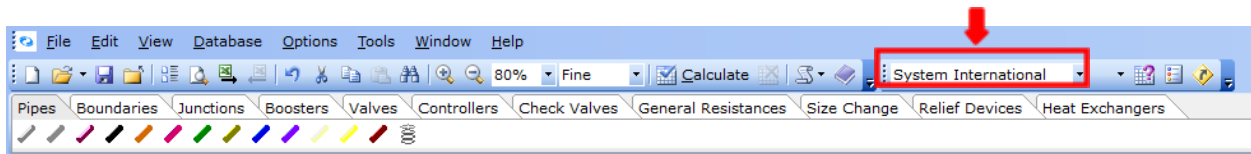


Figure 20: Unit Selector.

18. Click the drop-down menu to select a unit set. There are two default unit sets available: System International and US Basic (Figure 21).

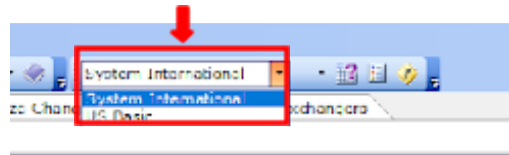


Figure 21: Unit Selector Drop-Down Menu.

19. You can click and select the desired unit set for use, or even create your own custom unit set. In this example, we will use System International and create our own unit set.

20. We can change our units by right mouse-clicking on the *Results* tab followed by selecting *Results Units* from the drop-down menu (Figure 22).

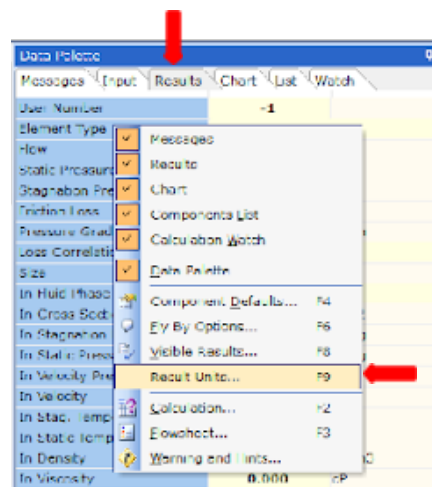


Figure 22: Result Units.

21. Alternatively, you can select F9. This opens a new dialogue box per Figure 23.

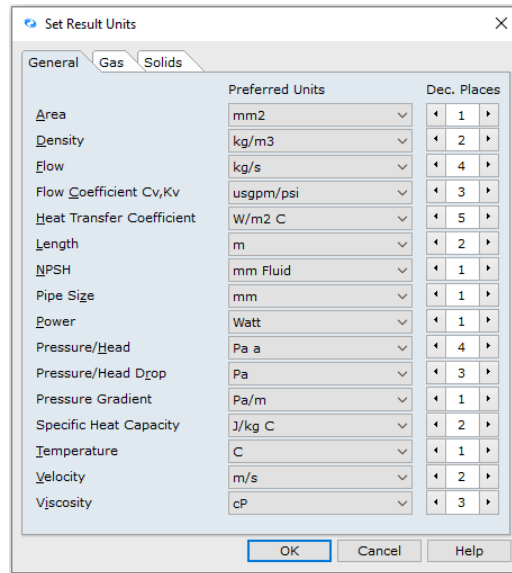


Figure 23: Set Result Units Dialogue Box.

22. Let us select the following units using the drop-down menu:

In this unit...	Select...
Flow	kg/h
Pressure/Head	bar g
Pressure/Head Drop	bar

23. The Set Result Units dialog box should now look like the screenshot given below (Figure 24). Select **OK** to apply the changes.

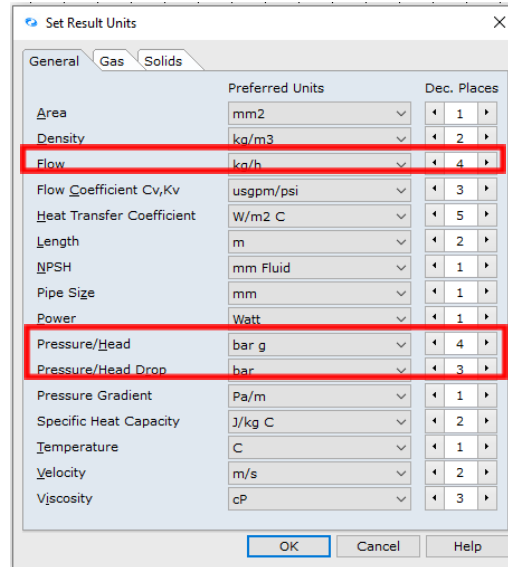


Figure 24: Modified Set Result Units Dialogue Box.

24. Now that we have set the results unit, let us save the custom result unit set. Click the drop-down button at the right of the default unit set and select the Save Environment option (Figure 25).

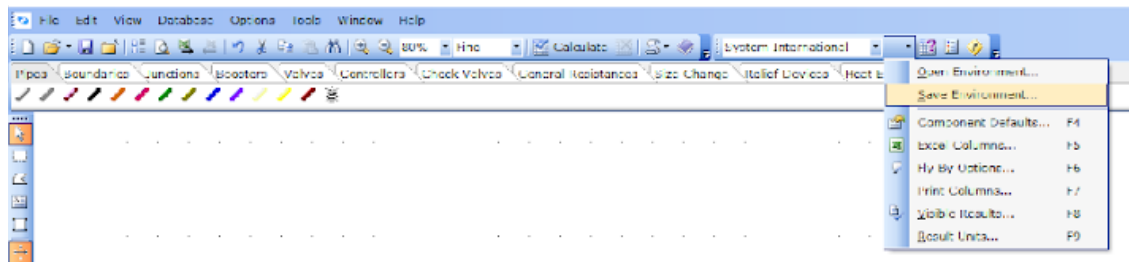


Figure 25: Save Environment Option.

25. The Save Environment dialogue box will appear (Figure 26).

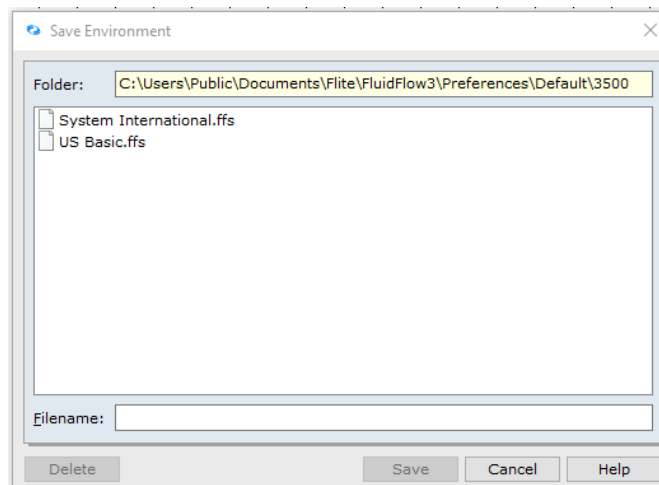


Figure 26: Save Environment Dialogue Box.

26. On the *Filename:* section type **Custom Unit 1** and click Save (Figure 27).

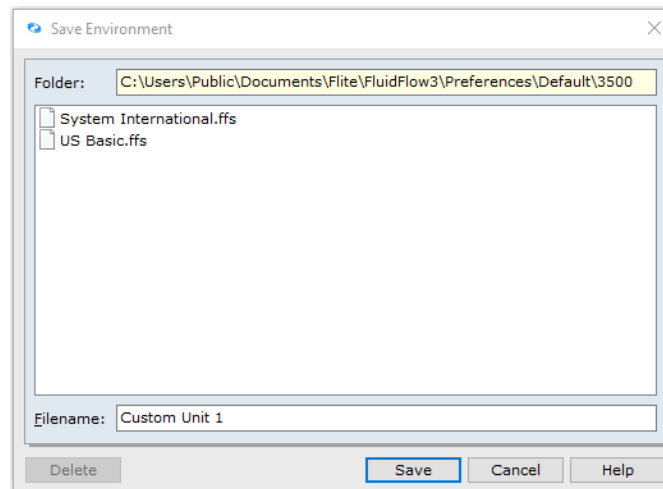


Figure 27: Modified Save Environment Dialog Box.

27. Now that we have successfully added our custom unit, it is now available in the unit set drop-down menu (Figure 28). Let us now use **Custom Unit 1** as our preferred unit set for this example.

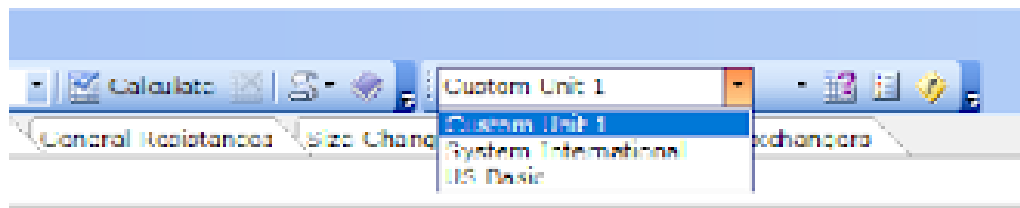


Figure 28: Unit Set Drop-Down Menu.

28. **Save** your flowsheet.

Task 4 – Calculate and Perform Hydraulic Analysis

Now that we have completed all the required inputs and customised the results unit set, we can now calculate the hydraulic model.

29. Calculate the model using the *Calculate* button located at the top of the flowsheet (Figure 29):

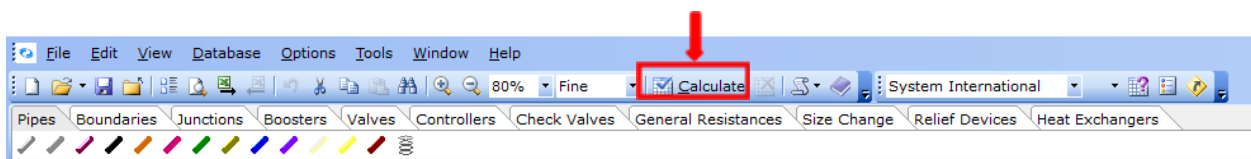


Figure 29: Calculate Button.

30. Check the converged hydraulic model in the flowsheet. The blue arrowhead shows that the flow direction is from the higher pressure node 1 to the lower pressure node 2. Also, check if the pipe is highlighted in **RED**, indicating that we have a warning message associated with the pipe element (Figure 30).

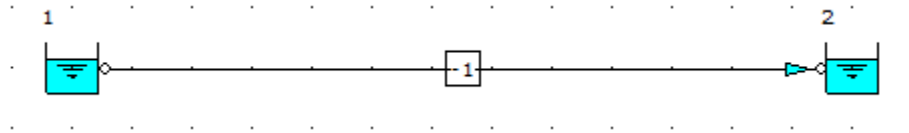


Figure 30: Converged Hydraulic Model

Note: Make sure that the **RED** Show or Hide Warnings Highlight button is selected in the *Flowsheet Toolbar* (Figure 31):

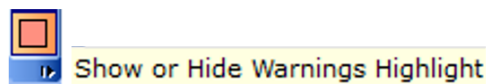


Figure 31: Show or Hide Warnings Highlight Button.

31. Select the *Messages* tab on the *Data Palette*, we can view all warnings for the model including a description of each (Figure 32). The converged hydraulic model has no warnings. Therefore, we can now proceed with the analysis of the results.

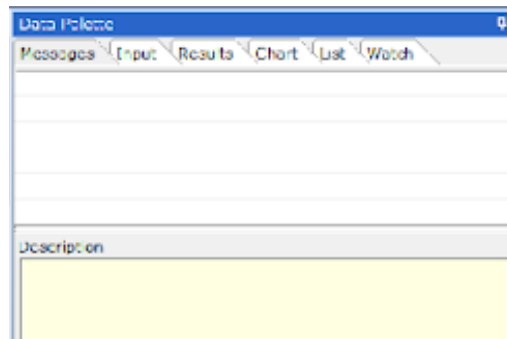


Figure 32: Warning Messages Tab.

Note: Warnings are enunciated by the software automatically to help the user eliminate any unwanted operating conditions and to prompt the user to develop a more efficient system design. Warnings should always be reviewed and considered by the engineer. You can evaluate and choose to ignore warnings if you wish.

32. Left-click the pipe number -1 and select the *Results* tab on the *Data Palette*. We can see all the calculated values for this pipe. Figure 33 provides an overview of these results.

Data Palette		
Messages	Input	Results
User Number	-1	
Element Type	Steel Pipe, Duct or Tube	
Flow	3488.6140	kg/h
Flow at STP	2846.4	m3/h
Flow at NTP	2697.6	m3/h
Static Pressure Loss	0.101	bar
Stagnation Pressure Loss	0.100	bar
Friction Loss	0.100	bar
Pressure Gradient	500.0	Pa/m
Loss Correlation	Duxbury	
Economic Velocity	9.26	m/s
Exact Economic Size	232.1	mm
Size	102.3	mm
In Fluid Phase	Gas or Vapor	
In Cross Section Flow Area	8219.4	mm2
In Stagnation Pressure	1.1000	bar g
In Static Pressure	1.0716	bar g
In Velocity Pressure	0.028	bar
In Velocity	48.17	m/s
In Mach Number	0.14	
In Stag. Temperature	25.0	C
In Static Temperature	23.8	C
In Density	2.45	kg/m3
In Viscosity	0.018	cP
In Specific Heat Capacity	1007.47	J/kg C
Joule-Thomson Coefficient	0.385	C/bar
Out Fluid Phase	Gas or Vapor	
Out Cross Section Flow Area	8219.4	mm2
Out Stagnation Pressure	1.0000	bar g
Out Static Pressure	0.9702	bar g
Out Velocity Pressure	0.030	bar
Out Velocity	50.48	m/s
Out Mach Number	0.15	
Out Stag. Temperature	25.0	C
Out Static Temperature	23.7	C
Out Density	2.34	kg/m3
Out Viscosity	0.018	cP
Out Specific Heat Capacity	1007.45	J/kg C
Composition Mass %	air	100.0%
Reynolds No	662829.1	
Friction Factor	0.017371	

- Suggested Economic Pipe Ø.
- Actual I.D. of 4" Sch. 40 Pipe.

The "In" values represent the calculated conditions at the pipe Inlet.

The "Out" values represent the calculated conditions at the pipe Outlet.

Figure 33: Calculated Results.

33. Warnings are enunciated based on the settings defined in *Warnings & Hints*. You can view these set-points by selecting; Options | Warnings & Hints or alternatively, select the *Warnings & Hints* icon at the top of the flowsheet.

Warning & Hints icon:

Since we are modeling a gas system, we are only interested in the Gas Limits (Figure 34).

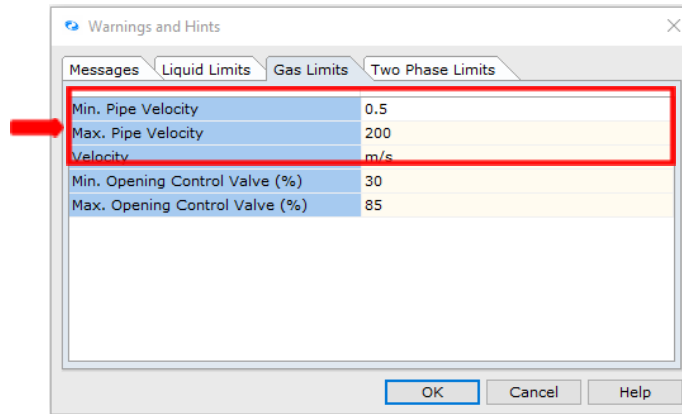


Figure 34: Warning and Hints: Gas Limits.

34. As we can see in Figure 33, the actual flowing velocities of the fluid are **48.17 m/s (inlet)** and **50.48 m/s (outlet)** which is within the **0.5 – 200 m/s** range for the minimum and maximum pipe velocity (Figure 34). The high-velocity warning is enunciated once the actual flowing velocity is higher than the maximum velocity level set in our warnings (**200 m/s** – See Figure 34).

35. FluidFlow automatically calculates an *Economic Velocity* and associated *Exact Economic Size* for each pipe in the model. These results can be viewed on the *Results* tab (Figure 33) and are provided as a suggestion in order to help develop an efficient system design. Note, it is down to the engineer's discretion as to whether or not the actual pipe size needs to be changed to be more in line with that suggested by the software. Therefore, the results for *Economic Velocity* and associated *Exact Economic Size* are suggested values only and do not have any effect on the overall operating conditions of the system.

36. We have established the pressure loss across the pipe element in this system and that the total stagnation pressure loss will be:

Stagnation Pressure at inlet boundary (Node 1):	1.10 barg.
Stagnation Pressure at outlet boundary (Node 2):	1.00 barg.
Stagnation Pressure Loss:	0.100 bar.

We know this is correct since we had an inlet pressure of **1.10 barg** and outlet pressure of **1.00 barg** which gives us a difference of **0.100 bar** (Figure 33).

37. To answer the problem in this example, the following required parameters are also calculated by the FluidFlow:

Parameter	Value
Flowrate	3489 kg/h
Reynolds Number	662829
Darcy Friction Factor	0.017371
Friction loss	0.100 bar

38. Note that we can improve the presentation of our model by selecting the pipe and changing the *Draw Thickness* and *Draw Color* from the Input Inspector. We have set the *Draw Thickness* to **3** and *Draw Color* to **clRed** (Figure 35).

Data Palette	
Messages Input Results Chart List Watch	
Unique Name	
Status	On
Length	20
Length Unit	m
Geometry	Cylindrical
Use Database Size	Yes
Nominal Size	4 inch
Classification	Schedule 40
Friction Model	Moody
Use Database Roughness	Yes
Roughness Description	Clean or new
Use Database Scaling	No
Scaling (0 to 50%)	0
Sizing Model	Economic Velocity
Heat Loss Model	Ignore Heat Loss/Gain
Draw Thickness [1..5]	3
Draw Color	clRed
Properties on Flowsheet	Hide

Figure 35: Modified Draw Thickness and Draw Color.

39. The hydraulic model should now look like the screenshot given below (Figure 36).

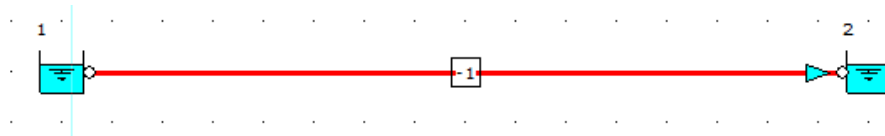


Figure 36: Hydraulic Model with Modified Draw Thickness and Draw Color.

40. Now let us show the properties of the boundaries. Left mouse-click on the inlet boundary (node 1) and whilst holding the SHIFT key, left mouse-click on the outlet boundary (node 2), and from the *Input Inspector*, change the field titled *Properties on Flowsheet* from *Hide* to *Show* (Figure 37).

Data Palette	
Messages Input Results Chart List Watch	
Unique Name	
Status	On
Elevation	0
Elevation Unit	m
Pressure Model	Stagnation Pressure
Pressure	1
Pressure Unit	bar g
Temperature	15
Temperature Unit	C
Fluid	water
Fluid Type	Newtonian/NIN-NonSetting
Properties on Flowsheet	Show
Alignment	Bottom
Font	Verdana,9,clWindowText,[]
Properties	

Figure 37: Show Properties on Flowsheet.

41. Now we will select the field titled *Properties* from the Input list. An *Element Properties* dialogue box will appear (Figure 38):

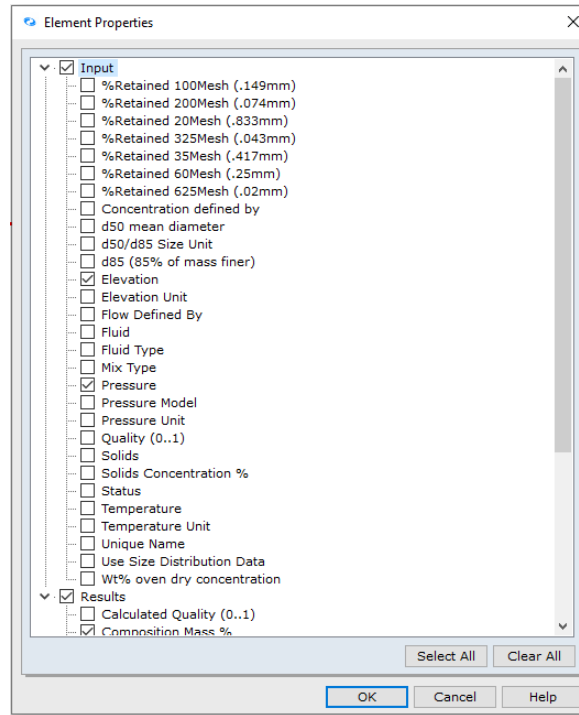


Figure 38: Element Properties Dialogue Box.

42. Place a checkmark in the box for the following parameters and click **OK**:

Element Properties	
Input	Results
Elevation	Composition %
Pressure	Density
	Temperature
	Viscosity

43. The hydraulic model should now look like the screenshot given below (Figure 39).

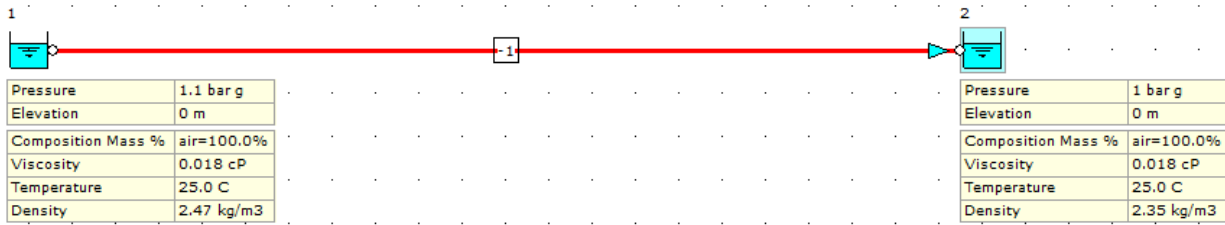


Figure 39: Hydraulic Model with Inlet and Outlet Boundary Properties.

44. Now let us show the properties of the pipe. Left mouse-click on the pipe number “-1” and from the *Input Inspector*, change the field titled *Properties on Flowsheet* from *Hide* to *Show* (Figure 40).

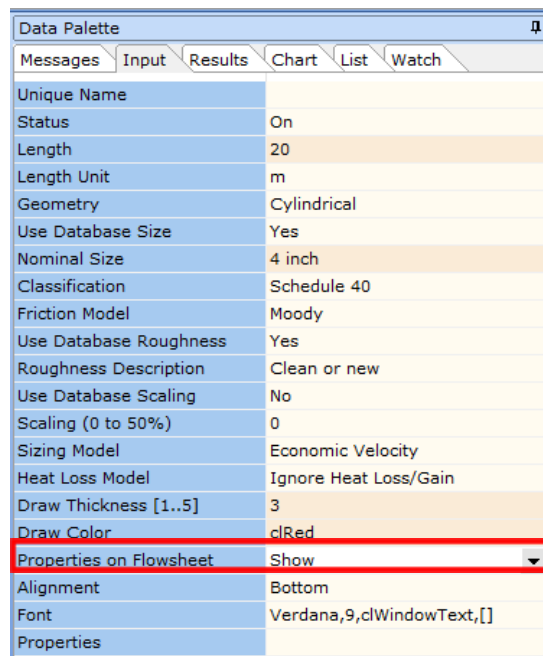


Figure 40: Show Properties on Flowsheet.

45. Now we will select the field titled *Properties* from the Input list. An *Element Properties* dialogue box will appear (Figure 41):

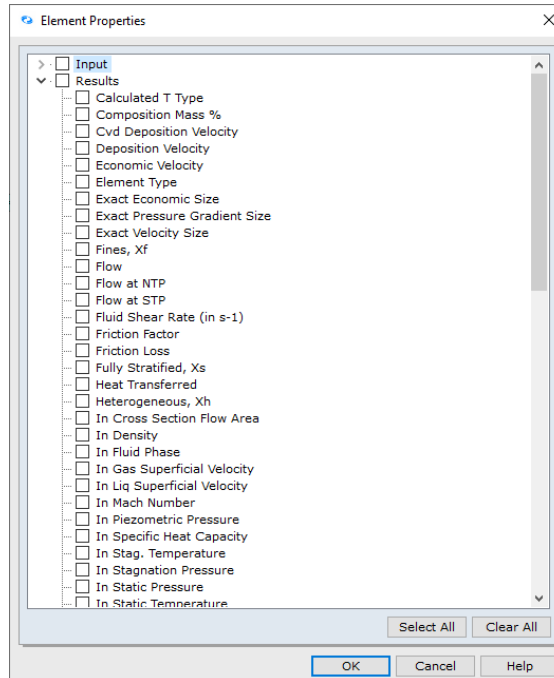


Figure 41: Element Properties Dialogue Box.

46. Place a checkmark in the box for the following parameters and click **OK**:

Element Properties	
Input	Results
Length Nominal Size	Flow Friction Factor Friction Loss In Density In Velocity In Viscosity Out Density Out Velocity Out Viscosity Reynolds No Size

47. The hydraulic model should now look like the screenshot given below (Figure 42).

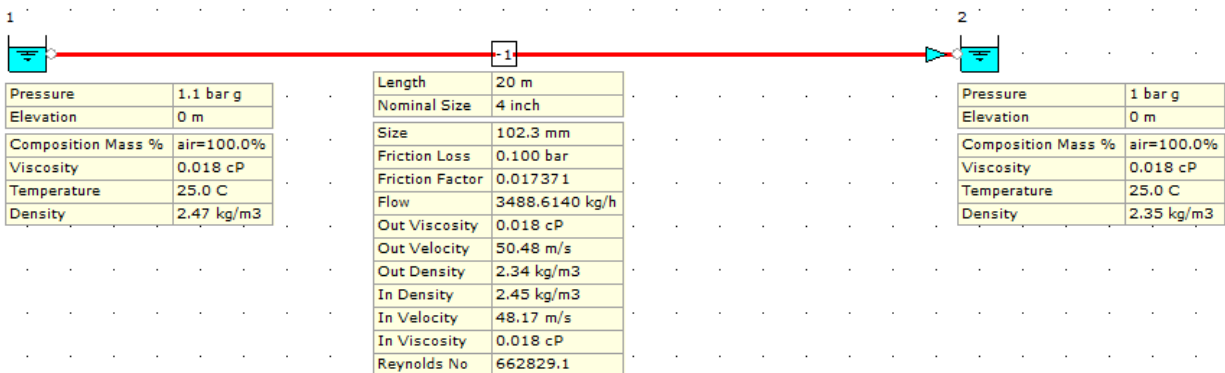


Figure 42: Hydraulic Model with Pipe Properties.

48. **Save** your flowsheet.

Task 5 – Compare the FluidFlow Results to Hand Calculation

FluidFlow software is designed to allow the modeling of fluid behaviour within complex piping systems, and accurately predict how the system will work for a given set of design conditions.

In this task, we will verify the accuracy of FluidFlow against our hand calculation.

Given:

$$P_1 = 1.1 \text{ barg} = 110000 \text{ Pa g} = 211325 \text{ Pa a}$$

$$P_2 = 1.0 \text{ barg} = 100000 \text{ Pa g} = 201325 \text{ Pa a}$$

4" SCH 40

$$L = 20.0 \text{ m}$$

$$\text{Air at } T = 25^\circ\text{C} = 298.15 \text{ K}$$

Required:

$$\text{Mass Flow Rate, } \dot{m} = ?$$

$$\text{Inlet Velocity, } V_{in} = ?$$

$$\text{Outlet Velocity, } V_{out} = ?$$

$$\text{Inlet Volumetric Flow Rate, } \dot{q}_{in} = ?$$

$$\text{Outlet Volumetric Flow Rate, } \dot{q}_{out} = ?$$

Reynolds Number, $Re = ?$

Darcy Friction Factor, $f_D = ?$

Friction loss = ?

Solution:

$$z_1 = 0 \text{ m}$$

$$z_2 = 0 \text{ m}$$

$$4" \text{ SCH } 40, \text{ I.D.} = 102.3 \text{ mm} = 0.1023 \text{ m}$$

Molecular Weight of Air from the FluidFlow Properties Database

$$M_w = 28.9505 \frac{\text{g}}{\text{gmol}} = 28.9505 \times 10^{-3} \frac{\text{kg}}{\text{gmol}}$$

Density of Air at 25.0°C and 1.1 barg from the FluidFlow Calculated Properties Database

$$\rho_{Air_{in}} = 2.47 \frac{\text{kg}}{\text{m}^3}$$

Density of Air at 25.0°C and 1.0 barg from the FluidFlow Calculated Properties Database

$$\rho_{Air_{out}} = 2.35 \frac{\text{kg}}{\text{m}^3}$$

Viscosity of Air at 25.0°C, 1.1 barg, and 1.0 barg from the FluidFlow Calculated Properties Database

$$\mu_{Air_{in}} = \mu_{Air_{out}} = 0.018 \text{ cP}$$

Let us assume the following for this calculation:

- 1) *The flow is steady and one – dimensional.*
- 2) *Negligible gravitational effects and potential energy term.*
- 3) *There is no shaft work.*
- 4) *Assume skin frictional losses only.*
- 5) *The fluid is an ideal gas of constant specific heat.*
- 6) *The flow of fluid in a conduit of constant cross – section is kept isothermal.*

Let us now solve for the value of the Reynolds number as a function of mass velocity G . Since G is still unknown let us represent it as variable.

$$Re = \frac{DV\rho}{\mu} = \frac{DG}{\mu}$$

$$Re = \frac{(0.1023 \text{ m})(G)}{0.018 \text{ cP} \times \frac{\text{Pa}\cdot\text{s}}{1000 \text{ cP}}}$$

$$Re = \frac{17050}{3}G \rightarrow \text{Equation 1}$$

Perry's Chemical Engineer's Handbook – 8th ed.

The Colebrook formula Equation (6 – 38)

$$\frac{1}{\sqrt{f_F}} = -4 \log \log \left[\frac{\epsilon}{3.7D} + \frac{1.256}{Re\sqrt{f_F}} \right] \quad Re > 4,000$$

Perry's Chemical Engineer's Handbook – 8th ed.

Table 6 – 1 Value of Surface Roughness for Various Materials

Commercial steel or wrought iron, $\epsilon = 0.0457 \text{ mm}$

Substitute the values of D , ϵ , and Re from Equation 1.

$$\frac{1}{\sqrt{f_F}} = -4 \log \log \left[\frac{0.0457 \text{ mm}}{3.7(102.3 \text{ mm})} + \frac{1.256}{\frac{17050}{3}G\sqrt{f_F}} \right] \rightarrow \text{Equation 2}$$

Perry's Chemical Engineer's Handbook – 8th ed.

Isothermal Gas Flow in Pipes and Channels Equation (6 – 114)

$$P_1^2 - P_2^2 = G^2 \frac{RT}{M_w} \left[\frac{4fL}{D_H} + 2 \ln \ln \left(\frac{P_1}{P_2} \right) \right]$$

Left – Hand Side Equation:

$$P_1^2 - P_2^2 = (211325 \text{ Pa a})^2 - (201325 \text{ Pa a})^2 = 4,126,500,000$$

Right – Hand Side Equation:

$$G^2 \frac{RT}{M_w} \left[\frac{4fL}{D_H} + 2 \ln \ln \left(\frac{P_1}{P_2} \right) \right] = G^2 \frac{\left(8.314 \frac{\text{m}^3\text{-Pa}}{\text{gmol}\cdot\text{K}}\right)(298.15 \text{ K})}{28.9505 \times 10^{-3} \frac{\text{kg}}{\text{gmol}}} \left[\frac{4f_F(20 \text{ m})}{0.1023 \text{ m}} + 2 \ln \ln \left(\frac{211325 \text{ Pa a}}{201325 \text{ Pa a}} \right) \right]$$

$$G^2 \frac{RT}{M_w} \left[\frac{4fL}{D_H} + 2 \ln \ln \left(\frac{P_1}{P_2} \right) \right] = 85,622.66973G^2 \left[782.0137f_F + 2 \ln \ln \left(\frac{8453}{8053} \right) \right]$$

Combine Left – Hand Side and Right – Hand Side Equations:

$$4,126,500,000 = 85,622.66973G^2 \left[782.0137f_F + 2 \ln \ln \left(\frac{8453}{8053} \right) \right]$$

$$48,194.01232 = G^2 \left[782.0137f_F + 2 \ln \ln \left(\frac{8453}{8053} \right) \right] \rightarrow \text{Equation 3}$$

Trial and Error (Iteration)

Trial 1

Step 1: Assume the value of G and solve for the value of f_F using Equation 2.

$$\frac{1}{\sqrt{f_F}} = -4 \log \log \left[\frac{0.0457 \text{ mm}}{3.7(102.3 \text{ mm})} + \frac{1.256}{\frac{17050}{3}G\sqrt{f_F}} \right] \rightarrow \text{Equation 2}$$

$$\text{Assume } G = 100 \frac{\text{kg}}{\text{m}^2\text{-s}}$$

$$\frac{1}{\sqrt{f_F}} = -4 \log \log \left[\frac{0.0457 \text{ mm}}{3.7(102.3 \text{ mm})} + \frac{1.256}{\frac{17050}{3} \left(100 \frac{\text{kg}}{\text{m}^2\text{-s}} \right) \sqrt{f_F}} \right]$$

$$f_F = 4.3027 \times 10^{-3}$$

Step 2: Substitute the calculated value of the f_F to Equation 3.

$$48,194.01232 = G^2 \left[782.0137f_F + 2 \ln \ln \left(\frac{8453}{8053} \right) \right] \rightarrow \text{Equation 3}$$

$$48,194.01232 = G^2 \left[782.0137(4.3027 \times 10^{-3}) + 2 \ln \ln \left(\frac{8453}{8053} \right) \right]$$

$$G = 117.9914 \frac{\text{kg}}{\text{m}^2\text{-s}}$$

Step 3: Compare the assumed value of the G to the calculated value.

$$100 \neq 117.9914$$

$$G_{\text{assumed}} \neq G_{\text{calculated}}$$

Repeat Steps 1 – 3 until the value of $G_{\text{assumed}} = G_{\text{calculated}}$

Trial 2:

Step 1: Use the calculated value of G from the previous trial and solve for the value of f_F using Equation 2.

$$\frac{1}{\sqrt{f_F}} = -4 \log \log \left[\frac{0.0457 \text{ mm}}{3.7(102.3 \text{ mm})} + \frac{1.256}{\frac{17050}{3} G \sqrt{f_F}} \right] \rightarrow \text{Equation 2}$$

$$\text{Assume } G = 117.9914 \frac{\text{kg}}{\text{m}^2 \cdot \text{s}}$$

$$\frac{1}{\sqrt{f_F}} = -4 \log \log \left[\frac{0.0457 \text{ mm}}{3.7(102.3 \text{ mm})} + \frac{1.256}{\frac{17050}{3} \left(117.9914 \frac{\text{kg}}{\text{m}^2 \cdot \text{s}} \right) \sqrt{f_F}} \right]$$

$$f_F = 4.2704 \times 10^{-3}$$

Step 2: Substitute the calculated value of f_F to Equation 3.

$$48,194.01232 = G^2 \left[782.0137 f_F + 2 \ln \ln \left(\frac{8453}{8053} \right) \right] \rightarrow \text{Equation 3}$$

$$48,194.01232 = G^2 \left[782.0137 (4.2704 \times 10^{-3}) + 2 \ln \ln \left(\frac{8453}{8053} \right) \right]$$

$$G = 118.4242 \frac{\text{kg}}{\text{m}^2 \cdot \text{s}}$$

Step 3: Compare the assumed value of G to the calculated value.

$$117.9914 \neq 118.4242$$

$$G_{\text{assumed}} \neq G_{\text{calculated}}$$

Repeat Steps 1 – 3 until the value of $G_{\text{assumed}} = G_{\text{calculated}}$

Trial 3:

Step 1: Use the calculated value of G from the previous trial and solve for the value of f_F using Equation 2.

$$\frac{1}{\sqrt{f_F}} = -4 \log \log \left[\frac{0.0457 \text{ mm}}{3.7(102.3 \text{ mm})} + \frac{1.256}{\frac{17050}{3} G \sqrt{f_F}} \right] \rightarrow \text{Equation 2}$$

$$\text{Assume } G = 118.4242 \frac{\text{kg}}{\text{m}^2 \cdot \text{s}}$$

$$\frac{1}{\sqrt{f_F}} = -4 \log \log \left[\frac{0.0457 \text{ mm}}{3.7(102.3 \text{ mm})} + \frac{1.256}{\frac{17050}{3} \left(118.4242 \frac{\text{kg}}{\text{m}^2 \cdot \text{s}} \right) \sqrt{f_F}} \right]$$

$$f_F = 4.2697 \times 10^{-3}$$

Step 2: Substitute the calculated value of f_F to Equation 3.

$$48,194.01232 = G^2 \left[782.0137 f_F + 2 \ln \ln \left(\frac{8453}{8053} \right) \right] \rightarrow \text{Equation 3}$$

$$48,194.01232 = G^2 \left[782.0137 (4.2697 \times 10^{-3}) + 2 \ln \ln \left(\frac{8453}{8053} \right) \right]$$

$$G = 118.4337 \frac{kg}{m^2-s}$$

Step 3: Compare the assumed value of G to the calculated value.

$$118.4242 \neq 118.4337$$

$$G_{\text{assumed}} \neq G_{\text{calculated}}$$

Repeat Steps 1 – 3 until the value of $G_{\text{assumed}} = G_{\text{calculated}}$

Trial 4:

Step 1: Use the calculated value of G from the previous trial and solve for the value of f_F using Equation 2.

$$\frac{1}{\sqrt{f_F}} = -4 \log \log \left[\frac{0.0457 \text{ mm}}{3.7(102.3 \text{ mm})} + \frac{1.256}{\frac{17050}{3} G \sqrt{f_F}} \right] \rightarrow \text{Equation 2}$$

$$\text{Assume } G = 118.4337 \frac{kg}{m^2-s}$$

$$\frac{1}{\sqrt{f_F}} = -4 \log \log \left[\frac{0.0457 \text{ mm}}{3.7(102.3 \text{ mm})} + \frac{1.256}{\frac{17050}{3} \left(118.4337 \frac{kg}{m^2-s} \right) \sqrt{f_F}} \right]$$

$$f_F = 4.2697 \times 10^{-3}$$

Step 2: Substitute the calculated value of f_F to Equation 3.

$$48,194.01232 = G^2 \left[782.0137 f_F + 2 \ln \ln \left(\frac{8453}{8053} \right) \right] \rightarrow \text{Equation 3}$$

$$48,194.01232 = G^2 \left[782.0137 (4.2697 \times 10^{-3}) + 2 \ln \ln \left(\frac{8453}{8053} \right) \right]$$

$$G = 118.4337 \frac{kg}{m^2-s}$$

Step 3: Compare the assumed value of G to the calculated value.

$$118.4337 = 118.4337$$

$$G_{\text{assumed}} = G_{\text{calculated}}$$

Stop the iteration.

Summary of Iteration

Trial	G_{assumed}	$f_{F, \text{calculated}}$	$G_{\text{calculated}}$
	100	4.3027×10^{-3}	117.9914
2	117.9914	4.2704×10^{-3}	118.4242
3	118.4242	4.2697×10^{-3}	118.4337
4	118.4337	4.2697×10^{-3}	118.4337

Therefore

$$G = 118.4337 \frac{\text{kg}}{\text{m}^2 \cdot \text{s}}$$

Calculate for the value of Darcy friction factor, f_D

$$f_D = 4f_F = 4(4.2697 \times 10^{-3})$$

$$f_D = 0.0171$$

Calculate for the value of Reynolds Number, Re using Equation 1

$$Re = \frac{17050}{3} G \rightarrow \text{Equation 1}$$

$$Re = \frac{17050}{3} \left(118.4337 \frac{\text{kg}}{\text{m}^2 \cdot \text{s}} \right)$$

$$Re = 673098.195$$

Calculate for the value of mass flow rate, \dot{m}

$$\dot{m} = GS = 118.4337 \frac{\text{kg}}{\text{m}^2 \cdot \text{s}} \left[\frac{\pi}{4} (0.1023 \text{ m})^2 \right] \left[\frac{3600 \text{ s}}{1 \text{ h}} \right]$$

$$\dot{m} = 3504.4426 \frac{\text{kg}}{\text{h}}$$

Calculate for the value of inlet volumetric flow rate, \dot{q}_{in}

$$\dot{q}_{in} = \frac{GS}{\rho_{Air_{in}}} = \frac{118.4337 \frac{\text{kg}}{\text{m}^2 \cdot \text{s}} \left[\frac{\pi}{4} (0.1023 \text{ m})^2 \right] \left[\frac{3600 \text{ s}}{1 \text{ h}} \right]}{2.47 \frac{\text{kg}}{\text{m}^3}}$$

$$\dot{q}_{in} = 1418.8027 \frac{\text{m}^3}{\text{h}}$$

Calculate for the value of inlet volumetric flow rate, \dot{q}_{out}

$$\dot{q}_{out} = \frac{GS}{\rho_{Air_{out}}} = \frac{118.4337 \frac{kg}{m^2 \cdot s} \left[\frac{\pi}{4} (0.1023 \text{ m})^2 \right] \left[\frac{3600 \text{ s}}{1 \text{ h}} \right]}{2.35 \frac{kg}{m^3}}$$

$$\dot{q}_{out} = 1491.2522 \frac{m^3}{h}$$

Calculate for the value of inlet velocity, V_{in}

$$V_{in} = \frac{G}{\rho_{Air_{in}}} = \frac{118.4337 \frac{kg}{m^2 \cdot s}}{2.47 \frac{kg}{m^3}}$$

$$V_{in} = 47.9489 \frac{m}{s}$$

Calculate for the value of inlet velocity, V_{out}

$$V_{out} = \frac{G}{\rho_{Air_{out}}} = \frac{118.4337 \frac{kg}{m^2 \cdot s}}{2.35 \frac{kg}{m^3}}$$

$$V_{out} = 50.3973 \frac{m}{s}$$

Calculate for the value of average density, $\bar{\rho}$

$$\bar{\rho} = \frac{\rho_{in} + \rho_{out}}{2} = \frac{2.47 + 2.35}{2} = 2.41 \frac{kg}{m^3}$$

Calculate for the value of friction loss

$$\text{Friction Loss} = F_{Line} = h_{Line} = \frac{-\Delta P}{\rho} = \frac{4f_F L \bar{V}^2}{2Dg_c} = \frac{4f_F L \left(\frac{G}{\rho} \right)^2}{2Dg_c}$$

$$\text{Friction Loss} = -\Delta P = \frac{2f_F L G^2}{\rho D g_c}$$

$$\text{Friction Loss} = -\Delta P = \frac{2(4.2697 \times 10^{-3})(20 \text{ m}) \left(118.4337 \frac{kg}{m^2 \cdot s} \right)^2}{\left(2.41 \frac{kg}{m^3} \right) (0.1023 \text{ m}) \left(1 \frac{kg \cdot m}{N \cdot s^2} \right)} = 9717 \text{ Pa}$$

$$\text{Friction Loss} = -\Delta P = 9717 \text{ Pa} \times \frac{1.01325 \text{ bar}}{101325 \text{ Pa}}$$

$$\text{Friction Loss} = -\Delta P = 0.097 \text{ bar}$$

Results Comparison:

Description	Unit	FluidFlow Results	Hand Calculation	% Difference
	kg/h	3489	3504	0.4290%
<i>Inlet Velocity, V_{in}</i>	m/s	48.17	47.95	0.4578%
<i>Outlet Velocity, V_{out}</i>	m/s	50.48	50.40	0.1586%
<i>Inlet Vol. Flow Rate, \dot{q}_{in}</i>	m ³ /h	1411	1419	0.5654%
<i>Outlet Vol. Flow Rate, \dot{q}_{out}</i>	m ³ /h	1481	1491	0.6729%
<i>Reynolds Number, Re</i>	-	662829	673098	1.5374%
<i>Darcy Friction Factor, f_D</i>	-	0.0174	0.0171	1.7391%
<i>Friction loss</i>	bar	0.100	0.097	3.0457%

Commentary:

The results compare very well with the hand calculation.