FluidFlow

The Flow of Liquid via Pressure Difference

**Flite Software N.I. Ltd, Block E, Balliniska Business Park, Springtown Rd, Derry, BT48 0LY, N. Ireland. T: 44 (0) 2871 279227 |
E: sales@fluidflowinfo.com | www.fluidflowinfo.com**

FluidFlow

The Flow of Liquid 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 water at 25ºC. 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 liquid 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).

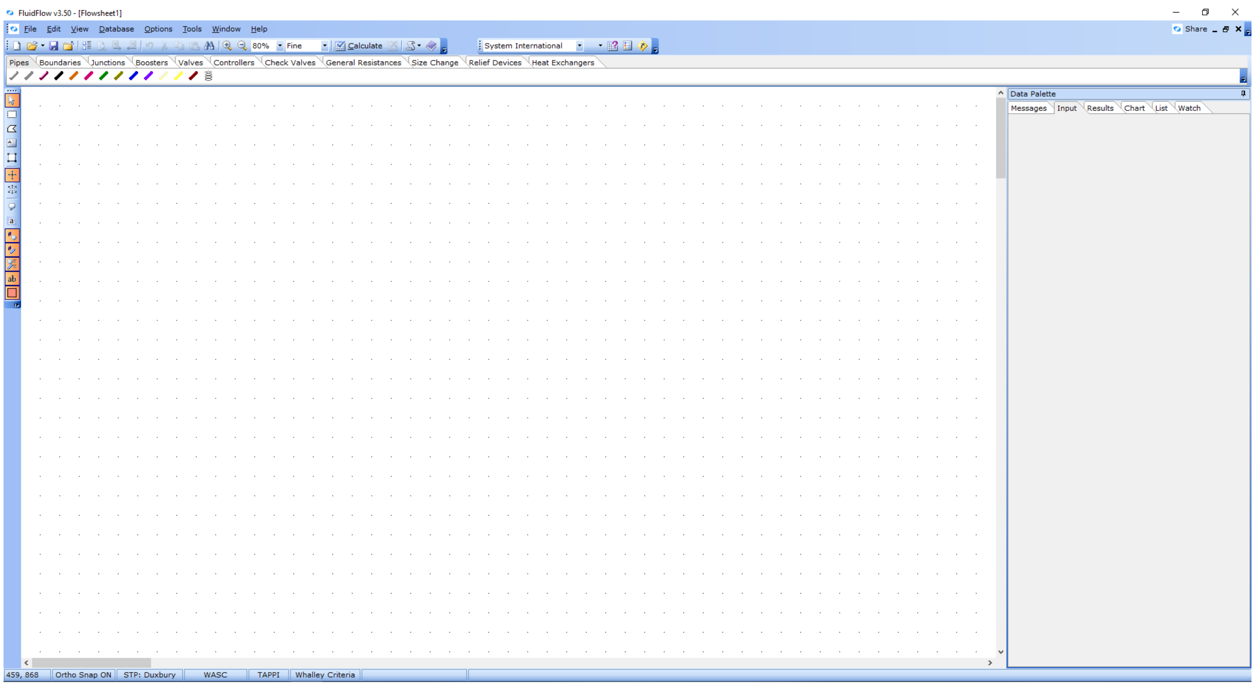

Figure 1: New flowsheet user interface.

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).

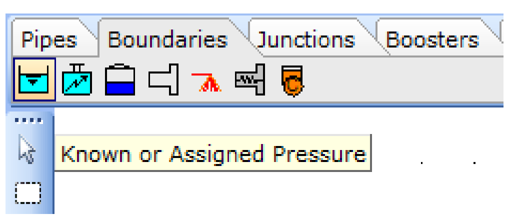

Figure 2: Known or Assigned Pressure Node.

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).

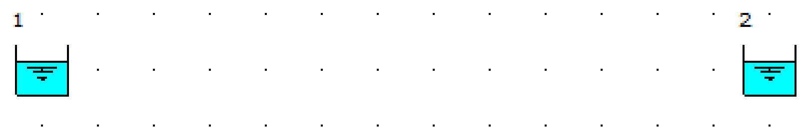

Figure 3: Known Pressure Nodes.

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).

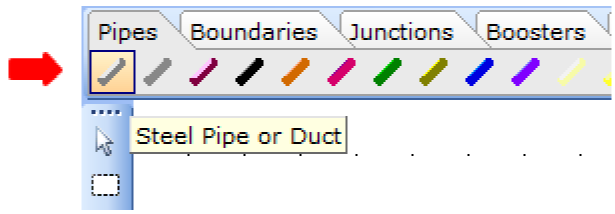

Figure 4: Steel Pipe of Duct Icon.

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.

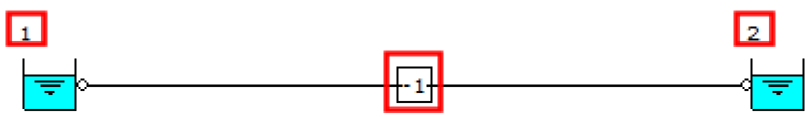

Figure 5: Hydraulic Model.

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):

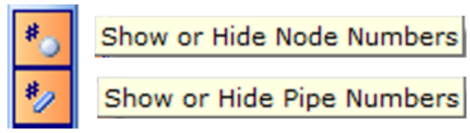

Figure 6: Show or Hide Node/Pipe Numbers buttons.

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 **01-FF Liq 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).

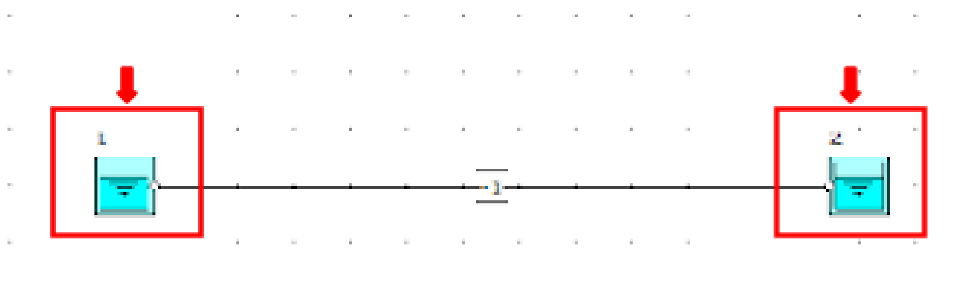

Figure 7: Highlighted Nodes.

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

FluidFlow

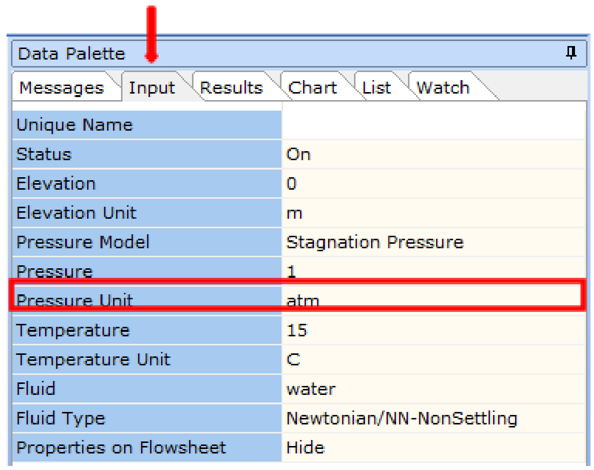

Figure 8: Input Inspector.

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.

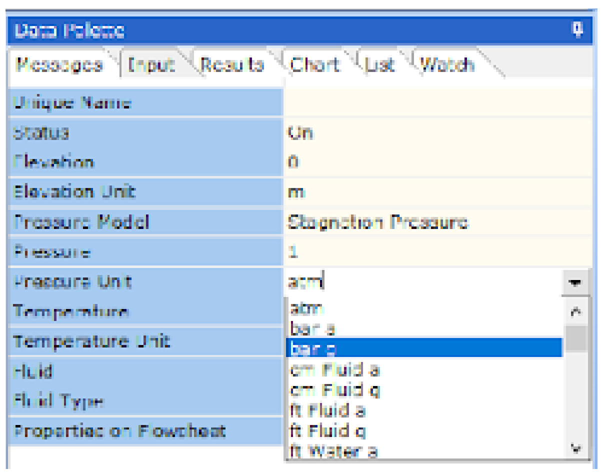

Figure 9: Pressure Units.

11. We can now change the inlet operating pressure and fluid temperature 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

The Input Inspector on the *Data Palette* should now look like the screenshot given below (Figure 10).

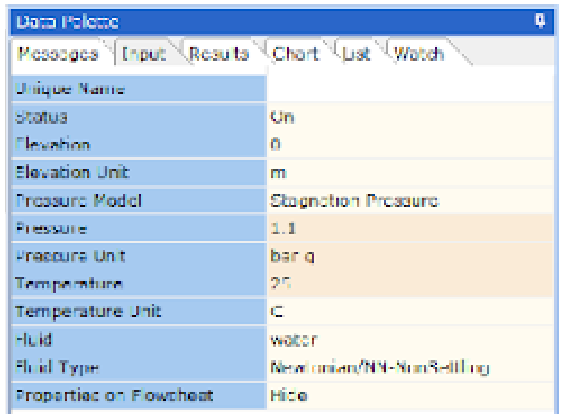

Figure 10: Input Inspector at Node 1.

Note that we will keep the default water as our **Fluid**.

12. Let us now check the input values in node 2 (Figure 11).

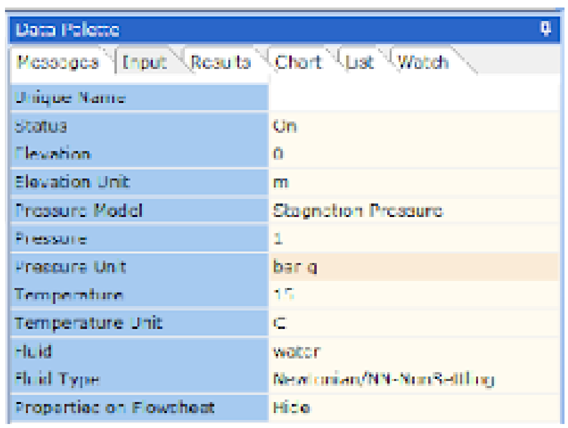

Figure 11: Input Inspector at Node 2.

The operating pressure is already at 1 barg which is consistent with our specification for node 2. For the temperature input, we don't need to change it to 25ºC. 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.

13. 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 12).

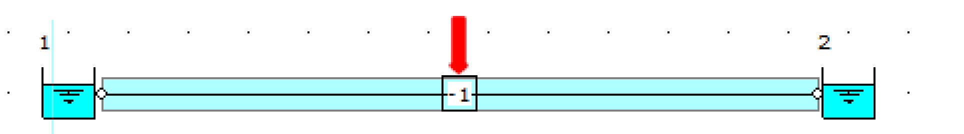

Figure 12: Highlighted Pipe.

14. 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 13):

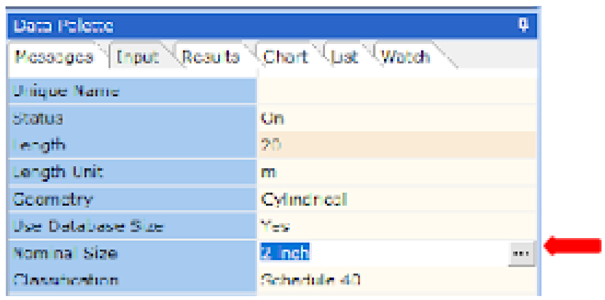

Figure 13: Nominal Size with 3 dots button.

Click the button and a *Set Pipe Dimension* dialog box will appear (Figure 14):

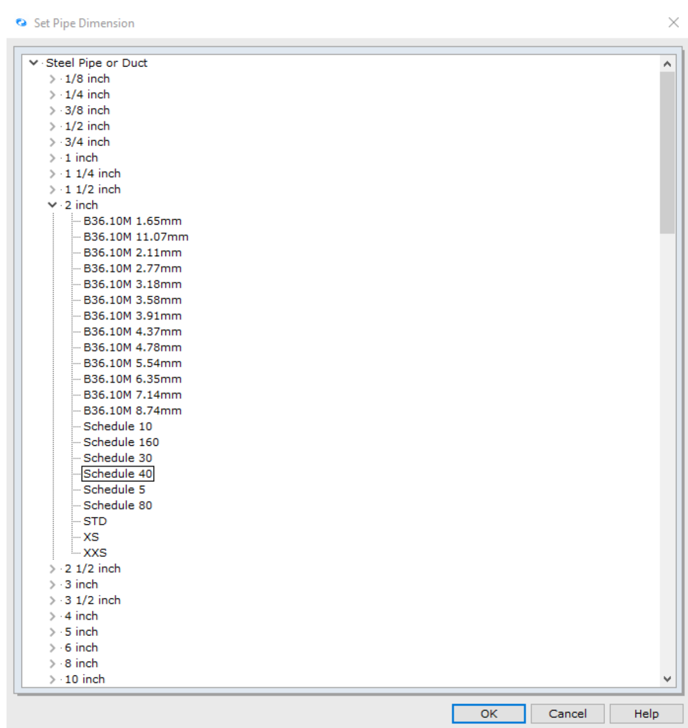

Figure 14: Set Pipe Dimension Dialog Box.

This dialog box will allow us to access the pipe database (Figure 14). 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 15).

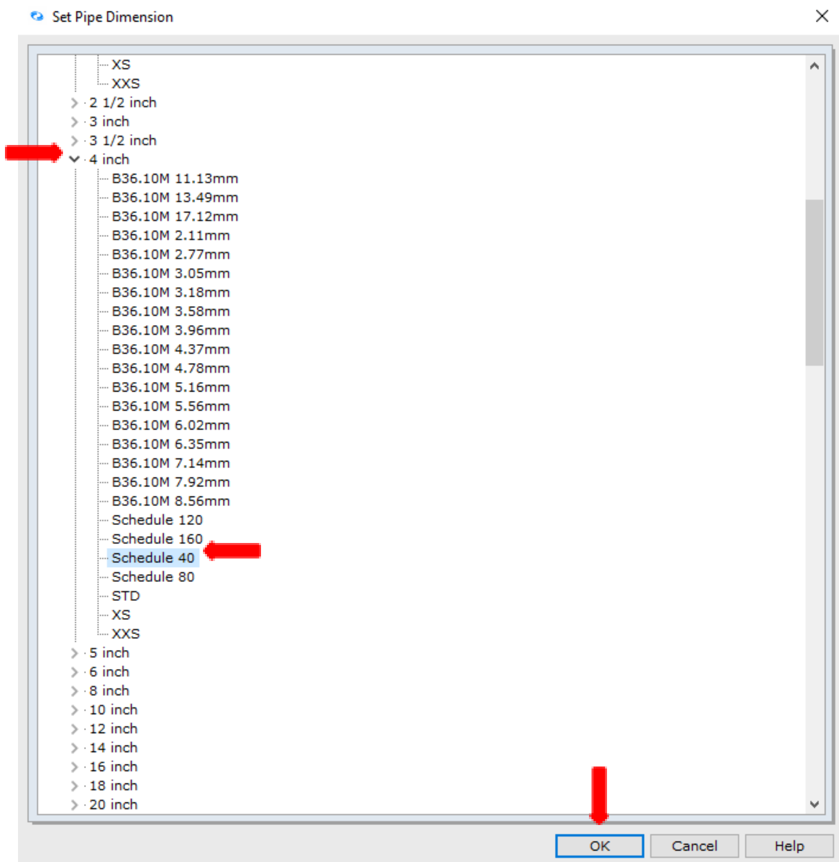

Figure 15: Set Pipe Dimension Dialog Box.

15. The Input Inspector on the *Data Palette* should now look like the screenshot given below (Figure 16).

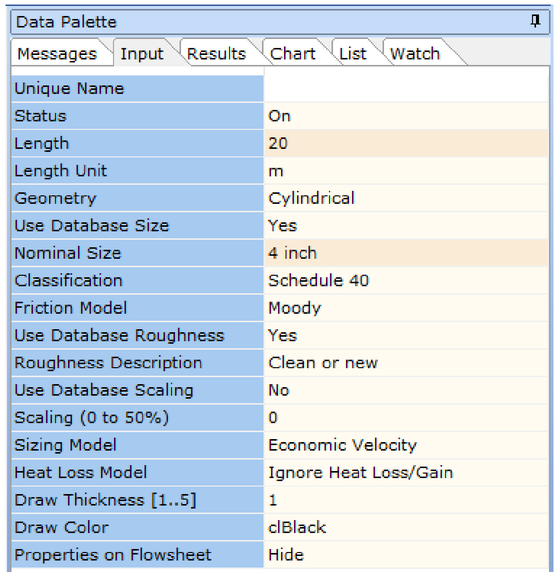

Figure 16: Input Inspector at Pipe Number "-1".

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 17):

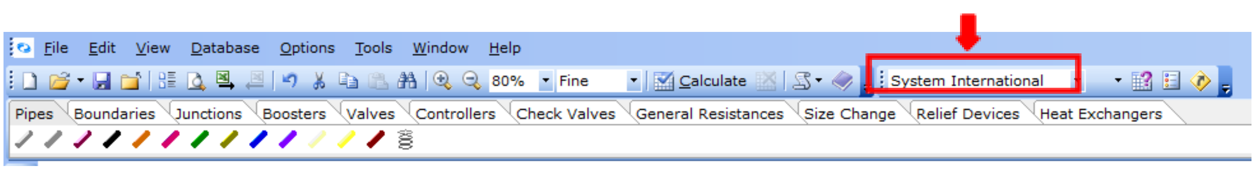

Figure 17: Unit Selector.

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

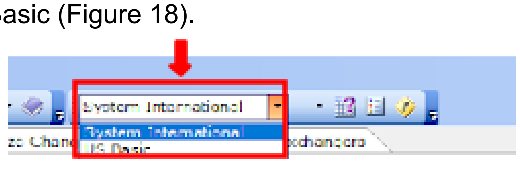

Figure 18: Unit Selector Drop-Down Menu.

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 19).

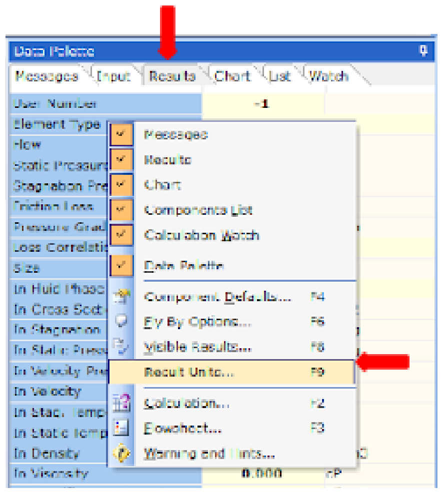

Figure 19: Result Units.

21. Alternatively, you can select F9. This opens a new dialogue box per Figure 20.

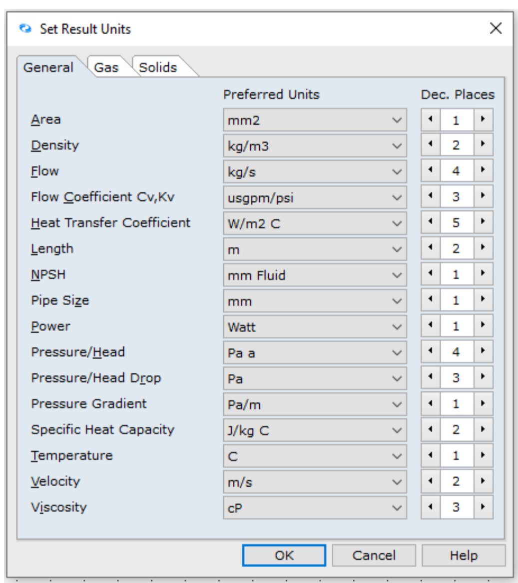

Figure 20: Set Result Units Dialogue Box.

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

In this unit…	Select…
Flow	m^3/h
Pressure/Head	bar g
Pressure/Head Drop	bar

23. The Set Result Units dialogue box should now look like the screenshot given below (Figure 21). Select **OK** to apply the changes.

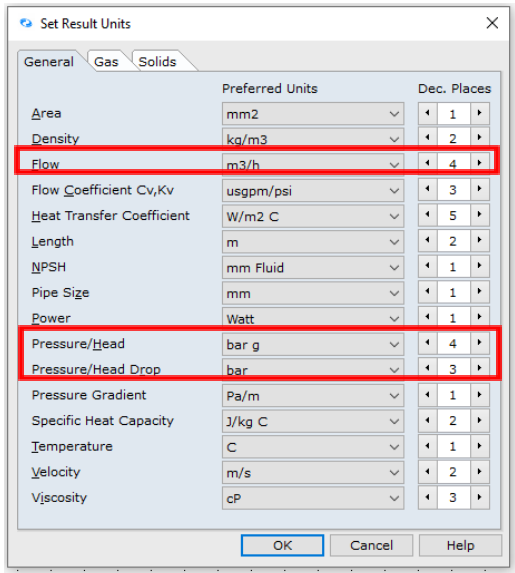

Figure 21: Modified Set Result Units Dialogue Box.

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 22).

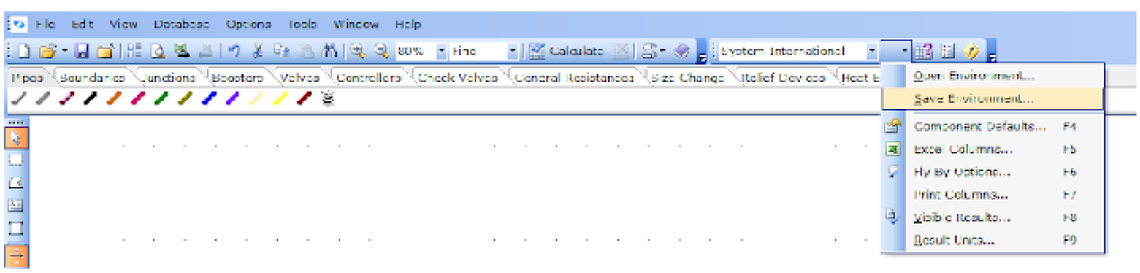

Figure 22: Save Environment Option.

25. The Save Environment dialogue box will appear (Figure 23).

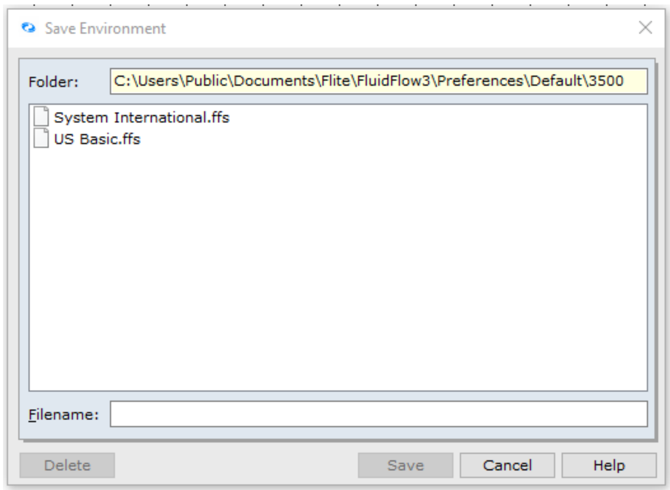

Figure 23: Save Environment Dialogue Box.

26. On the *Filename:* section type **Custom Unit 1** and click Save (Figure 24).

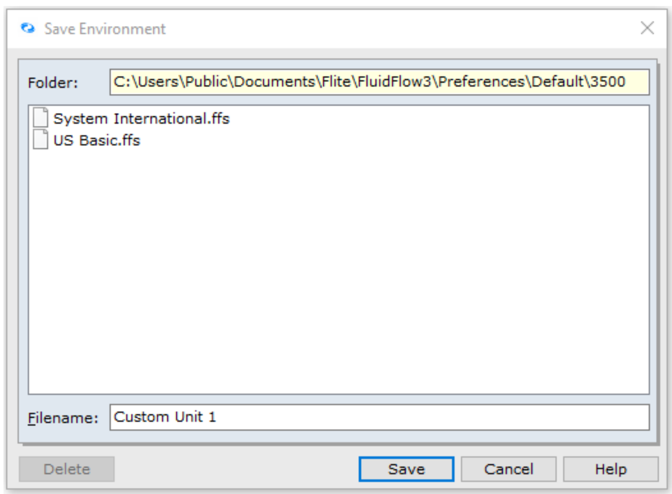

Figure 24: Modified Save Environment Dialogue Box.

27. Now that we have successfully added our custom unit, it is now available in the unit set drop-down menu (Figure 25). Let us now use **Custom Unit 1** as our preferred unit set for this example.

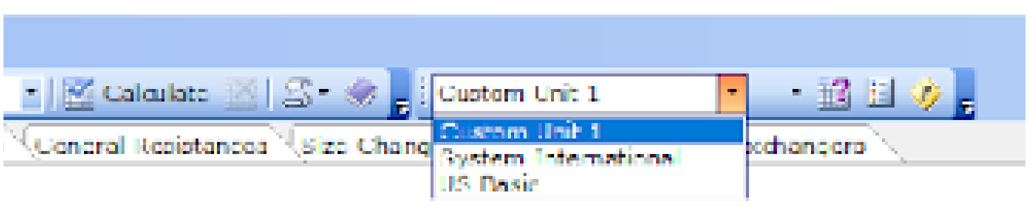

Figure 25: Unit Set Drop-Down Menu.

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 26):

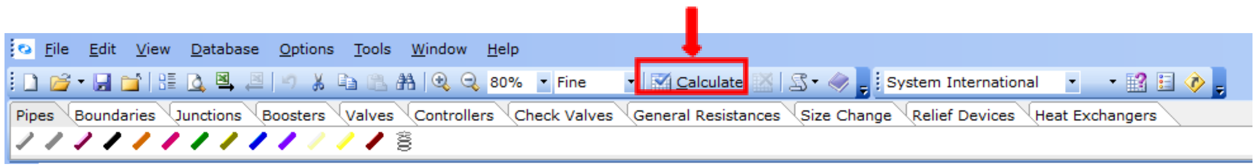

Figure 26: Calculate Button.

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 27).

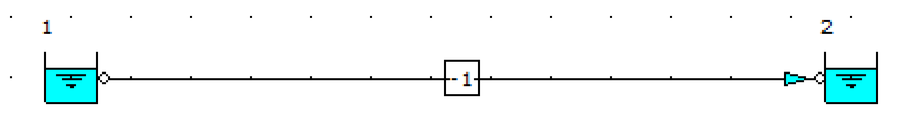

Figure 27: Converged Hydraulic Model

Note: Make sure that the **RED** Show or Hide Warnings Highlight button is selected in the *Flowsheet Toolbar* (Figure 28):

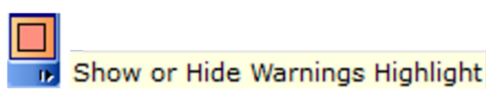

Figure 28: Show or Hide Warnings Highlight Button.

31. Select the *Messages* tab on the *Data Palette*, we can view all warnings for the model including a description of each (Figure 29). The converged hydraulic model has no warnings. Therefore, we can now proceed with the analysis of the results.

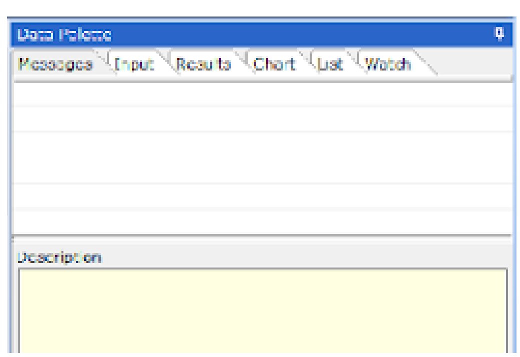

Figure 29: Warning Messages Tab.

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 30 provides an overview of these results.

Data Palette

Messages | Input | Results | Chart | List | Watch

User Number	-1	
Element Type	Steel Pipe, Duct or Tube	
Flow	70.0975	m3/h
Static Pressure Loss	0.100	bar
Stagnation Pressure Loss	0.100	bar
Friction Loss	0.100	bar
Pressure Gradient	500.0	Pa/m
Loss Correlation	Darcy	
Economic Velocity	1.25	m/s
Exact Economic Size	141.0	mm
Size	102.3	mm
In Fluid Phase	Liquid	
In Cross Section Flow Area	8219.4	mm2
In Piezometric Pressure	1.0720	bar g
In Stagnation Pressure	1.1000	bar g
In Static Pressure	1.0720	bar g
In Velocity Pressure	0.028	bar
In Velocity	2.37	m/s
In Stag. Temperature	25.0	C
In Static Temperature	25.0	C
In Density	997.10	kg/m3
In Viscosity	0.890	cP
In Specific Heat Capacity	4181.58	J/kg C
In Vapor Pressure	-0.9816	bar g
Out Fluid Phase	Liquid	
Out Cross Section Flow Area	8219.4	mm2
Out Piezometric Pressure	0.9720	bar g
Out Stagnation Pressure	1.0000	bar g
Out Static Pressure	0.9720	bar g
Out Velocity Pressure	0.028	bar
Out Velocity	2.37	m/s
Out Stag. Temperature	25.0	C
Out Static Temperature	25.0	C
Out Density	997.09	kg/m3
Out Viscosity	0.890	cP
Out Specific Heat Capacity	4181.61	J/kg C
Out Vapor Pressure	-0.9816	bar g
Composition Mass %	water	100.0%
Reynolds No	271490.7	
Friction Factor	0.018282	

- 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 30: 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 liquid system, we are only interested in the Liquid Limits (Figure 31).

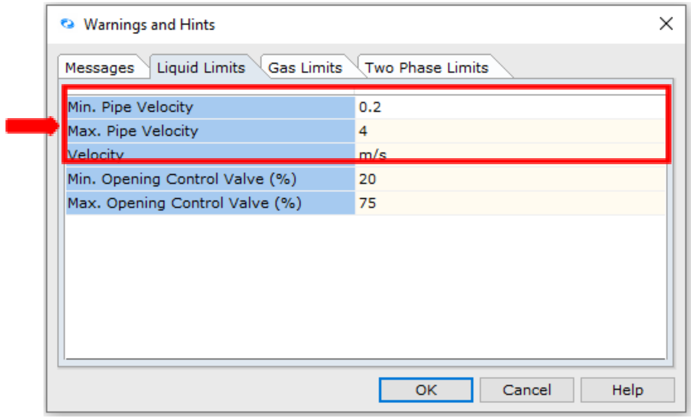

Figure 31: Warning and Hints: Liquid Limits.

34. As we can see in Figure 30, the actual flowing velocity of the fluid is **2.37 m/s** which is within the **0.2 – 4 m/s** range for the minimum and maximum pipe velocity (Figure 31). The high-velocity warning is enunciated once the actual flowing velocity is higher than the maximum velocity level set in our warnings (**4 m/s** – See Figure 31).

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 30) 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 30).

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

Parameter	Value
Flowrate	70 m^3/h
Reynolds Number	271491
Darcy Friction Factor	0.018282
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 to set the *Draw Thickness* to **3** and *Draw Color* to **clRed** (Figure 32).

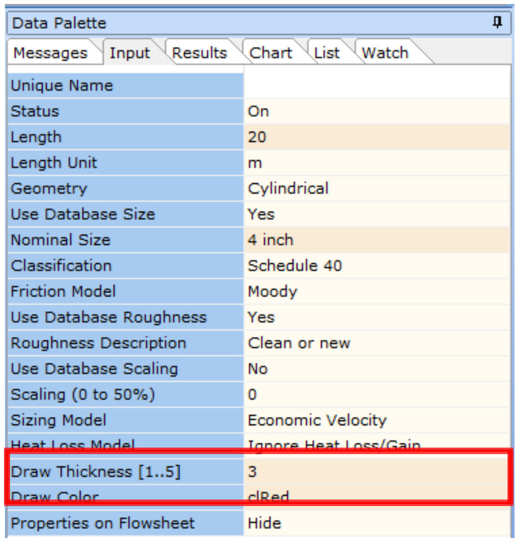

Figure 32: Modified Draw Thickness and Draw Color.

39. The hydraulic model should now look like the screenshot given below (Figure 33).

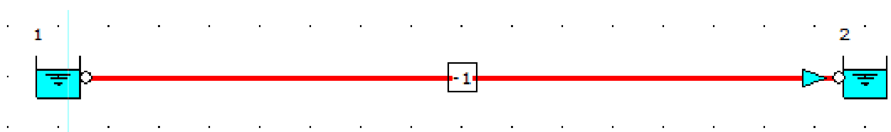

Figure 33: Hydraulic Model with Modified Draw Thickness and Draw Color.

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 34).

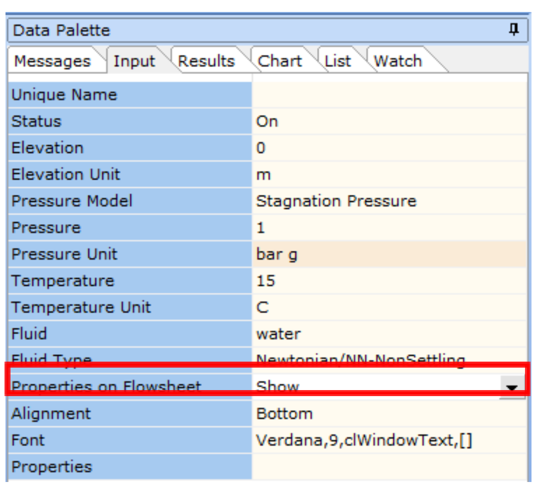

Figure 34: Show Properties on Flowsheet.

41. Now we will select the field titled *Properties* from the Input list. An *Element Properties* dialogue box will appear (Figure 35):

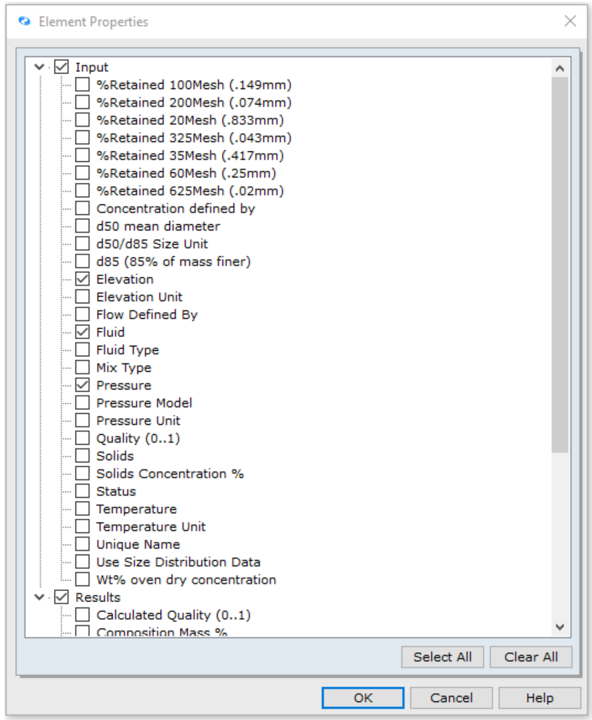

Figure 35: Element Properties Dialogue Box.

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

Element Properties	
Input	Results
Elevation Fluid Pressure	Density Temperature Viscosity

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

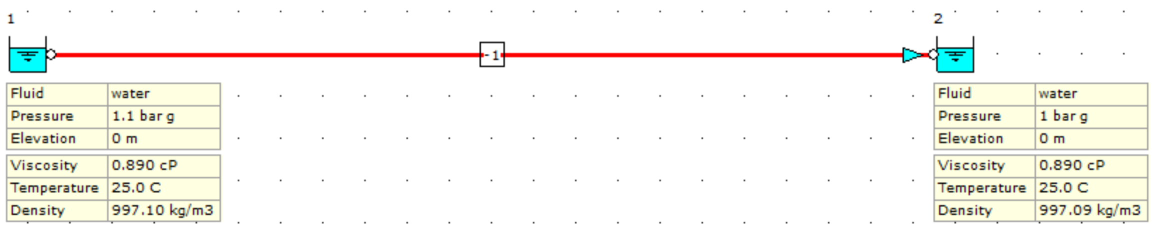

Figure 36: Hydraulic Model with Inlet and Outlet Boundary Properties.

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 37).

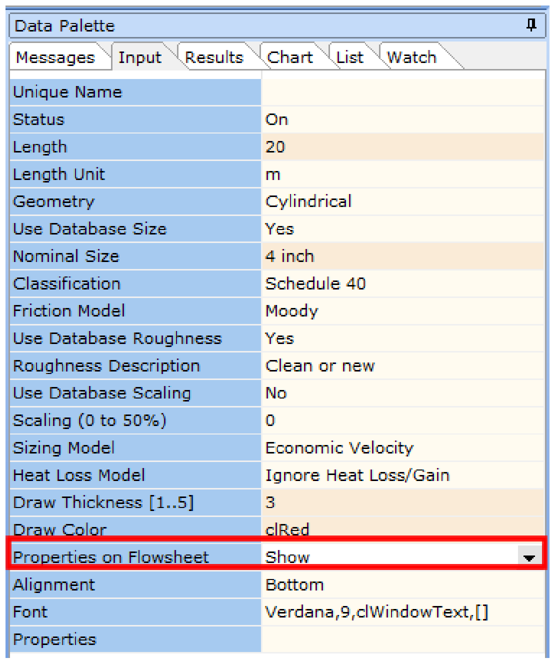

Figure 37: Show Properties on Flowsheet.

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

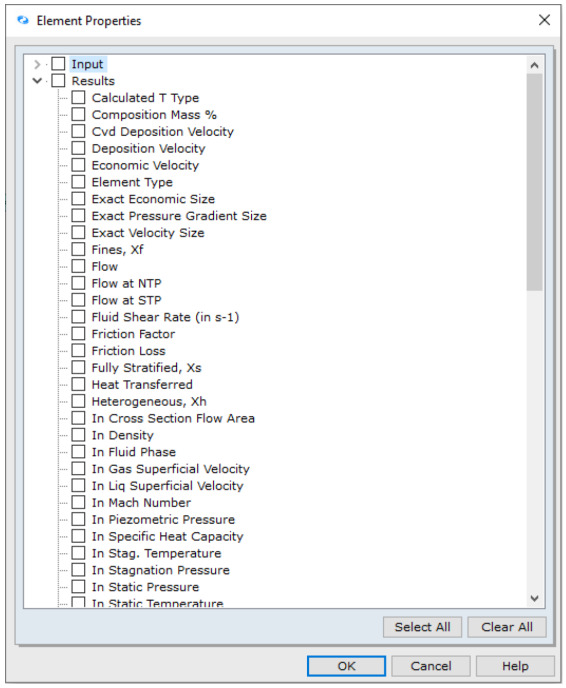

Figure 38: Element Properties Dialogue Box.

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 39).

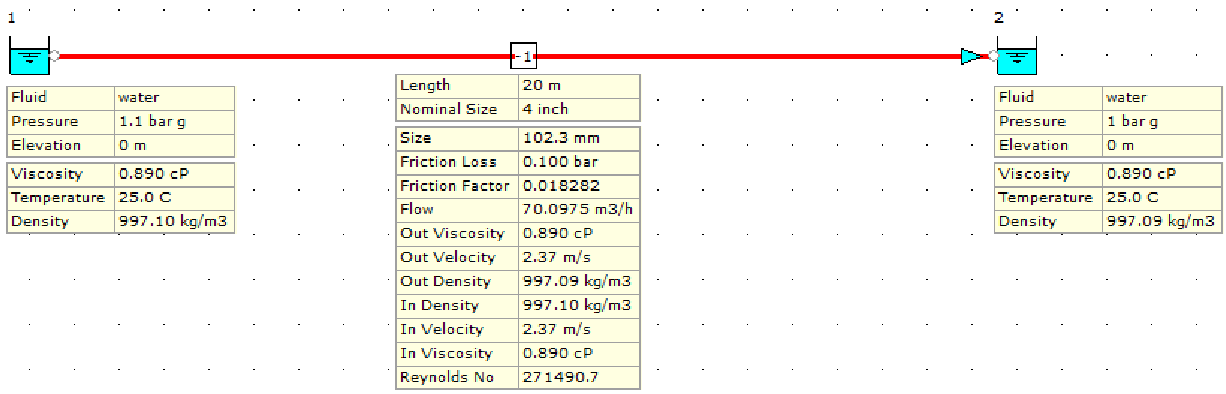

Figure 39: Hydraulic Model with Pipe Properties.

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.

$Given:$

$P_1 = 1.1\ barg$

$P_2 = 1.0\ barg$

$4"\ SCH\ 40$

$L = 20.0\ m$

$Water\ at\ 25°C$

$Required:$

$Volumetric\ Flow\ Rate,\ \dot{q} = \ ?$

$Velocity,\ V = \ ?$

$Reynolds\ Number,\ Re = \ ?$

$Darcy\ Friction\ Factor,\ f_D = \ ?$

$Friction\ loss = \ ?$

$Solution$:

$z_1 = 0\ m$

$z_2 = 0\ m$

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

$Density\ of\ Water\ at\ 25.0°C\ from\ FluidFlow\ Calculated\ Properties\ Database$

$\rho_{H_2O} = 997.10\ \frac{kg}{m^3}$

$Viscosity\ of\ Water\ at\ 25.0°C\ from\ FluidFlow\ Calculated\ Properties\ Database$

$\mu_{H_2O} = 0.890\ cP$

$Mechanical\ Energy\ Balance\ Equation$:

$$W^{'} = \Delta z\frac{g}{g_c} + \frac{\Delta V^2}{2g_c} + \frac{\Delta P}{\rho} + \sum F$$

$Summation\ of\ Frictional\ Losses$:

$$\sum F = F_{Line} + F_{Fittings} + F_{Contraction} + F_{Expansion} + F_{Metering}$$

$Let\ us\ simplify\ the\ following\ terms\ in\ the\ Mechanical\ Energy\ Balance\ Equation$:

$$W^{'} = 0\ (No\ pump)$$

$$\Delta z\frac{g}{g_c} = 0\ \left(Horizontal\ Pipe,\ z_1 = z_2\right)$$

$$\frac{\Delta V^2}{2g_c} = 0\ (Constant\ cross - sectional\ area,\ V_1 = V_2)$$

$$F_{Fittings} = F_{Contraction} = F_{Expansion} = F_{Metering} = 0$$

$$\sum F = F_{Line} = h_{Line} = \frac{4f_F LV^2}{2Dg_c}$$

$Applying\ the\ above\ conditions\ to\ Mechanical\ Energy\ Balance\ equation$:

$$0 = 0 + 0 + \frac{\Delta P}{\rho} + \frac{4f_F LV^2}{2Dg_c}$$

$$0 = \frac{P_2 - P_1}{\rho} + \frac{4f_F LV^2}{2Dg_c}$$

$$\frac{P_1 - P_2}{\rho} = \frac{2f_F L V^2}{D g_c}$$

$Since\ f_F\ and\ V\ are\ still\ unknown,\ let\ us\ arrange\ the\ equation\ and\ put\ it\ on\ the\ left-hand\ side$

$$f_F V^2 = \frac{D g_c (P_1 - P_2)}{2\rho L} = \frac{(0.1023\ m)\left(1\frac{kg-m}{N-s^2}\right)\left[(1.1-1)\ bar \times \frac{101325\ Pa}{1.01325\ bar}\right]}{2\left(997.10\ \frac{kg}{m^3}\right)(20.0\ m)}$$

$$f_F V^2 = 0.02564938321 \ \rightarrow Equation\ 1$$

$Let\ us\ now\ solve\ for\ the\ value\ of\ the\ Reynolds\ number.\ \ Since\ V\ is\ still\ unknown,\ let\ us\ represent\ it\ as\ variable.$

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

$$Re = \frac{(0.1023\ m)(V)\left(997.10\ \frac{kg}{m^3}\right)}{0.890\ cP \times \frac{Pa-s}{1000\ cP}}$$

$$Re = 114610.4831V \rightarrow Equation\ 2$$

$Perry's\ Chemical\ Engineer's\ Handbook - 8th\ ed.$

$The\ Colebrook\ Equation\ (6 - 38)$

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

$Perry's\ Chemical\ Engineer's\ Handbook - 8th\ ed.$

$Table\ 6 - 1\ Value\ of\ Surface\ Roughness\ for\ Various\ Materials$

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

$Substitute\ the\ values\ of\ D\ and\ \epsilon.$

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

$Trial\ and\ Error\ (Iteration)$

$Trial\ 1$

$Step\ 1: Assume\ the\ value\ of\ the\ f_F\ between\ 0.003 - 0.006\ and\ solve\ for\ the\ value\ of\ V$

$using\ Equation\ 1.$

$f_F V^2 = 0.02564938321 \rightarrow Equation\ 1$

$Assume\ f_F = 0.004$

$V = \sqrt{\frac{0.02564938321}{0.004}}$

$V = 2.5323\ \frac{m}{s}$

$Step\ 2: Substitute\ the\ calculated\ value\ of\ V\ to\ Equation\ 2.$

$Re = 114610.4831V \rightarrow Equation\ 2$

$Re = 114610.4831\left(2.5323\ \frac{m}{s}\right)$

$Re = 290228.1265$

$Step\ 3: Substitute\ the\ value\ of\ Re\ to\ Equation\ 3\ and\ calculate\ the\ value\ of\ f_F.$

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

$$\frac{1}{\sqrt{f_F}} = -\ 4 \log log\left[\frac{0.0457\ mm}{3.7(102.3\ mm)} + \frac{1.256}{290228.1265\sqrt{f_F}}\right]$$

$f_F = 4.4873\times10^{-3}$

$Step\ 4: Compare\ the\ assumed\ value\ of\ the\ f_F\ to\ the\ calculated\ value.$

$$0.004 \neq 4.4873\times10^{-3}$$

$$f_{F,\ assumed} \neq f_{F,\ calculated}$$

$Repeat\ Steps\ 1 - 4\ until\ the\ value\ of\ f_{F,\ assumed} = f_{F,\ calculated}$

$Trial\ 2:$

$Step\ 1: Use\ the\ calculated\ value\ of\ the\ f_F\ from\ the\ previous\ trial\ and\ solve\ for\ the\ value\ of\ V$

$using\ Equation\ 1.$

$f_F V^2 = 0.02564938321 \rightarrow Equation\ 1$

$Assume\ f_F = 4.4873\times10^{-3}$

$V = \sqrt{\frac{0.02564938321}{4.4873\times10^{-3}}}$

$V = 2.3908\ \frac{m}{s}$

$Step\ 2: Substitute\ the\ calculated\ value\ of\ V\ to\ Equation\ 2.$

$Re = 114610.4831V \rightarrow Equation\ 2$

$Re = 114610.4831\left(2.3908\ \frac{m}{s}\right)$

$Re = 274010.7431$

$Step\ 3: Substitute\ the\ value\ of\ Re\ to\ Equation\ 3\ and\ calculate\ the\ value\ of\ f_F.$

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

$$\frac{1}{\sqrt{f_F}} = -\ 4\log log\left[\frac{0.0457\ mm}{3.7(102.3\ mm)} + \frac{1.256}{274010.7431\sqrt{f_F}}\right]$$

$f_F = 4.5079\times10^{-3}$

$Step\ 4: Compare\ the\ assumed\ value\ of\ the\ f_F\ to\ the\ calculated\ value.$

$$4.4873\times10^{-3} \neq 4.5079\times10^{-3}$$

$$f_{F,\ assumed} \neq f_{F,\ calculated}$$

$Repeat\ Steps\ 1\ -\ 4\ until\ the\ value\ of\ f_{F,\ assumed} =\ f_{F,\ calculated}$

$Trial\ 3:$

$Step\ 1: Use\ the\ calculated\ value\ of\ f_F\ from\ the\ previous\ trial\ and\ solve\ for\ the\ value\ of\ V$

$using\ Equation\ 1.$

$f_F V^2 = 0.02564938321\ \rightarrow Equation\ 1$

$Assume\ f_F = 4.5079\times10^{-3}$

$V = \sqrt{\frac{0.02564938321}{4.5079\times10^{-3}}}$

$V = 2.3853\ \frac{m}{s}$

$Step\ 2: Substitute\ the\ calculated\ value\ of\ V\ to\ Equation\ 2.$

$Re\ =\ 114610.4831V \rightarrow Equation\ 2$

$Re\ =\ 114610.4831\left(2.3853\ \frac{m}{s}\right)$

$Re\ =\ 273380.3854$

$Step\ 3: Substitute\ the\ value\ of\ Re\ to\ Equation\ 3\ and\ calculate\ the\ value\ of\ f_F.$

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

$$\frac{1}{\sqrt{f_F}} =- 4\log log\left[\frac{0.0457\ mm}{3.7(102.3\ mm)} + \frac{1.256}{273380.3854\sqrt{f_F}}\right]$$

$f_F\ =\ 4.5087\times10^{-3}$

$Step\ 4: Compare\ the\ assumed\ value\ of\ the\ f_F\ to\ the\ calculated\ value.$

$$4.5079\times10^{-3}\neq4.5087\times10^{-3}$$

$$f_{F,\ assumed} \neq f_{F,\ calculated}$$

$Repeat\ Steps\ 1\ -\ 4\ until\ the\ value\ of\ f_{F,\ assumed}\ =\ f_{F,\ calculated}$

$Trial\ 4:$

$Step\ 1: Use\ the\ calculated\ value\ of\ the\ f_F\ from\ the\ previous\ trial\ and\ solve\ for\ the\ value\ of\ V$

$using\ Equation\ 1.$

$f_F V^2\ =\ 0.02564938321\ \rightarrow Equation\ 1$

$Assume\ f_F\ =\ 4.5087\times10^{-3}$

$$V\ =\sqrt{\frac{0.02564938321}{4.5087\times10^{-3}}}$$

$V\ =\ 2.3851\ \frac{m}{s}$

$Step\ 2: Substitute\ the\ calculated\ value\ of\ V\ to\ Equation\ 2.$

$Re\ =\ 114610.4831V \rightarrow Equation\ 2$

$Re = 114610.4831\left(2.3851\ \frac{m}{s}\right)$

$Re = 273357.4634$

$Step\ 3: Substitute\ the\ value\ of\ Re\ to\ Equation\ 3\ and\ calculate\ the\ value\ of\ f_F.$

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

$$\frac{1}{\sqrt{f_F}} =- \ 4\log log\left[\frac{0.0457\ mm}{3.7(102.3\ mm)} + \frac{1.256}{273357.4634\sqrt{f_F}}\right]$$

$f_F = 4.5088\times10^{-3}$

$Step\ 4: Compare\ the\ assumed\ value\ of\ the\ f_F\ to\ the\ calculated\ value.$

$$4.5087\times10^{-3}\neq4.5088\times10^{-3}$$

$$f_{F,\ assumed} \neq f_{F,\ calculated}$$

$Repeat\ Steps\ 1 - 4\ until\ the\ value\ of\ f_{F,\ assumed} = f_{F,\ calculated}$

$Trial\ 5:$

$Step\ 1: Use\ the\ calculated\ value\ of\ the\ f_F\ from\ the\ previous\ trial\ and\ solve\ for\ the\ value\ of\ V$

$using\ Equation\ 1.$

$f_F V^2 = 0.02564938321 \rightarrow Equation\ 1$

$Assume\ f_F = 4.5088\times10^{-3}$

$V = \sqrt{\frac{0.02564938321}{4.5088\times10^{-3}}}$

$V = 2.3851\ \frac{m}{s}$

$Step\ 2: Substitute\ the\ calculated\ value\ of\ V\ to\ Equation\ 2.$

$Re = 114610.4831V \rightarrow Equation\ 2$

$Re = 114610.4831\left(2.3851\ \frac{m}{s}\right)$

$Re = 273357.4634$

$Step\ 3: Substitute\ the\ value\ of\ Re\ to\ Equation\ 3\ and\ calculate\ the\ value\ of\ f_F.$

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

$$\frac{1}{\sqrt{f_F}} = -\ 4\log log\left[\frac{0.0457\ mm}{3.7(102.3\ mm)} + \frac{1.256}{273357.4634\sqrt{f_F}}\right]$$

$$f_F = 4.5088\times10^{-3}$$

$Step\ 4: Compare\ the\ assumed\ value\ of\ the\ f_F\ to\ the\ calculated\ value.$

$$4.5088\times10^{-3} = 4.5088\times10^{-3}$$

$$f_{F,\ assumed} = f_{F,\ calculated}$$

$Stop\ the\ iteration.$

$Summary\ of\ Iteration$

$Trial$	$f_{F,\ assumed}$	$V_{calculated}$	$Re_{calculated}$	$f_{F,\ calculated}$
	0.004	2.5323	290228.1265	4.4873×10^{-3}
2	4.4873×10^{-3}	2.3908	274010.7431	4.5079×10^{-3}
3	4.5079×10^{-3}	2.3853	273380.3854	4.5087×10^{-3}
4	4.5087×10^{-3}	2.3851	273357.4634	4.5088×10^{-3}
5	4.5088×10^{-3}	2.3851	273357.4634	4.5088×10^{-3}

$Therefore$

$$f_F = 4.5088\times10^{-3}$$

$$V = 2.3851\frac{m}{s}$$

$$Re = 273357.4634$$

$Calculate\ for\ the\ value\ of\ Darcy\ friction\ factor,\ f_D$

$$f_D = 4f_F = 4\left(4.5088\times10^{-3}\right)$$

$$f_D = 0.0180$$

$Calculate\ for\ the\ value\ of\ friction\ loss$

$$Friction\ Loss\ =\ F_{Line}\ =\ h_{Line}\ =\ \frac{-\Delta P}{\rho}\ =\ \frac{4f_F LV^2}{2Dg_c}$$

$$Friction\ Loss\ =-\ \Delta P\ =\ \frac{2f_F LV^2 \rho}{Dg_c}$$

$$Friction\ Loss\ =-\ \Delta P\ =\ \frac{2\left(4.5088\times10^{-3}\right)(20\ m)\left(2.3851\frac{m}{s}\right)^2\left(997.10\frac{kg}{m^3}\right)}{(0.1023\ m)\left(1\frac{kg-m}{N-s^2}\right)}\ =\ 10000\ Pa$$

$$Friction\ Loss\ =-\ \Delta P\ =\ 10000\ Pa\times\frac{1.01325\ bar}{101325\ Pa}$$

$$Friction\ Loss\ =-\ \Delta P\ =\ 0.100\ bar$$

$Calculate\ for\ the\ value\ of\ volumetric\ flow\ rate,\ \dot{q}$

$$\dot{q}\ =\ VS\ =\ V\left(\frac{\pi}{4}D^2\right)\ =\ 2.3851\frac{m}{s}\left[\left(\frac{\pi}{4}\right)(0.1023\ m)^2\right]\left[\frac{3600\ s}{1\ h}\right]$$

$$\dot{q}\ =\ 70.5749\frac{m^3}{h}$$

Results Comparison:

Description	Unit	FluidFlow Results	Hand Calculation	% Difference
Flow	m^3/h	70	71	1.4184%
Velocity	m/s	2.37	2.39	0.8403%
Reynolds number	-	271491	273357	0.6850%
Friction Factor	-	0.0183	0.0180	1.6529%
Friction Loss	bar	0.100	0.100	0%

Commentary:

The results compare very well with the hand calculation.