FluidFlow

The Flow of Gas via Elevation 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**

The Flow of Gas via Elevation 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 25ºC. The pipe inlet and outlet pressure shall be 1.0 barg. The pipe inlet elevation shall be 20 m and the outlet elevation of 10 m from grade.

Objective

The fluid will flow naturally from the source at a higher elevation to the sink at a lower elevation due to gravity (potential energy). Static head is the pressure resulting from a column of fluid acting under gravity. For compressible flow applications, changes in elevation are negligible compared to the other terms in the mechanical energy balance equation due to low-density values of gases.

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

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).

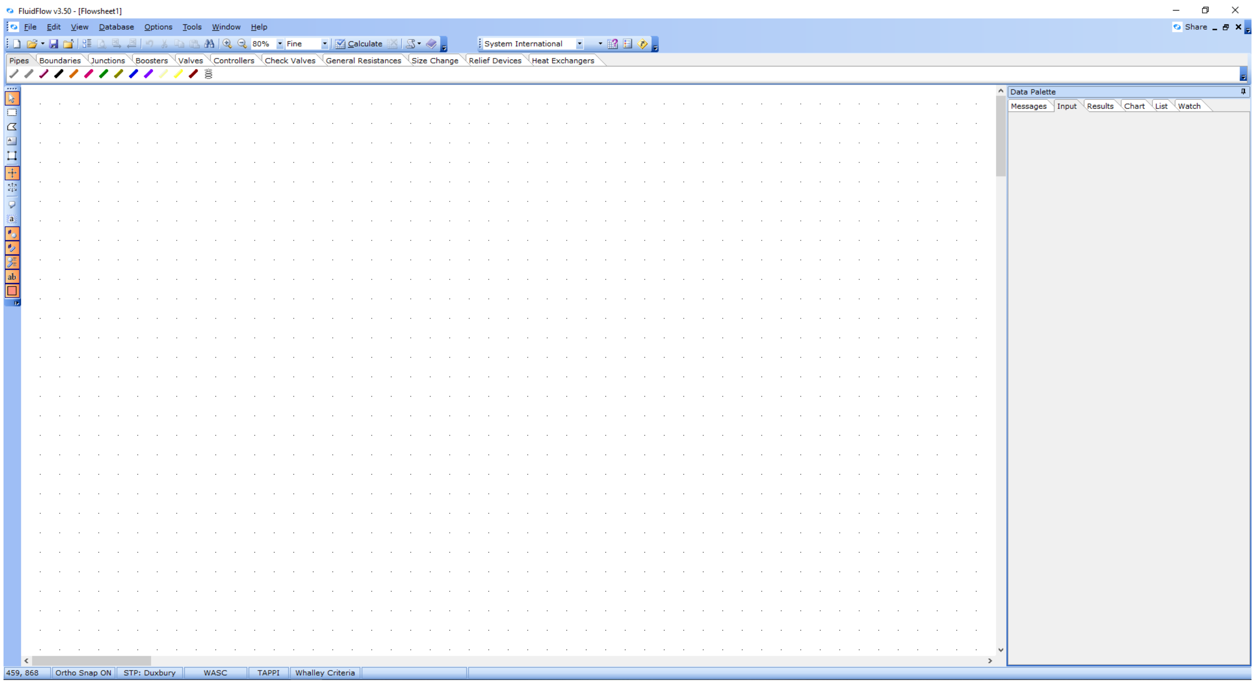


Figure 1: New flowsheet user interface.

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).

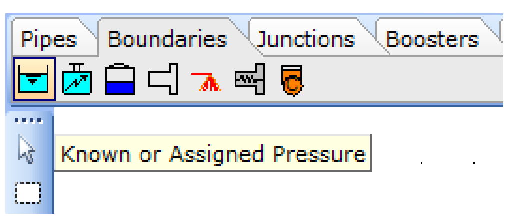


Figure 2: Known or Assigned Pressure Node.

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).

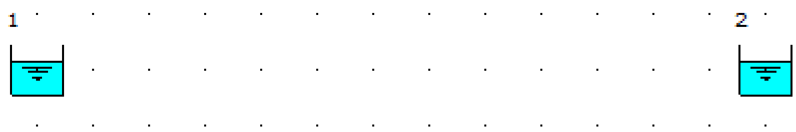


Figure 3: Known Pressure Nodes.

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).

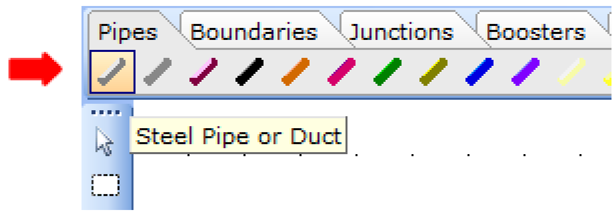


Figure 4: Steel Pipe of Duct Icon.

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.

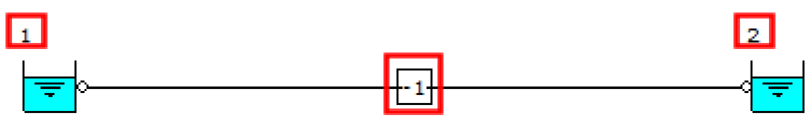


Figure 5: Hydraulic Model.

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):

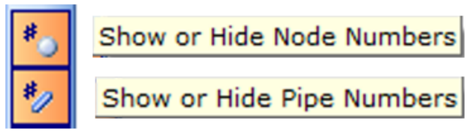


Figure 6: Show or Hide Node/Pipe Numbers buttons.

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 **05-FF Gas via Elevation 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).

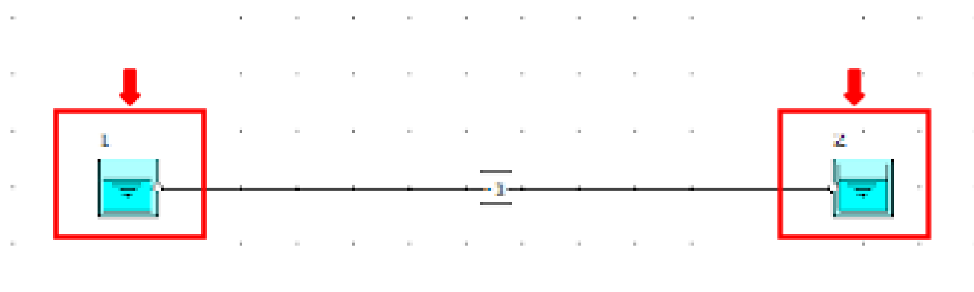


Figure 7: Highlighted Nodes.

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

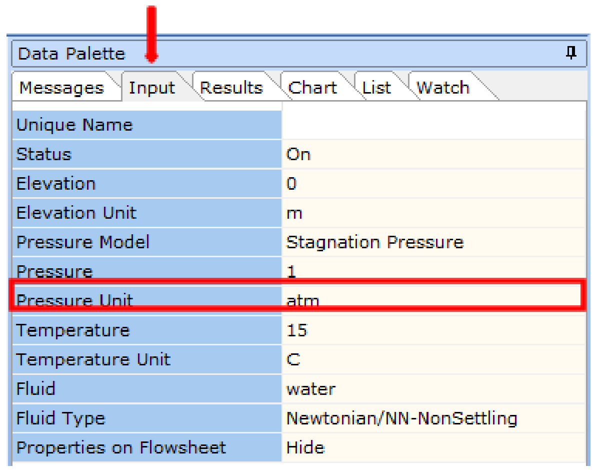


Figure 8: Input Inspector.

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.

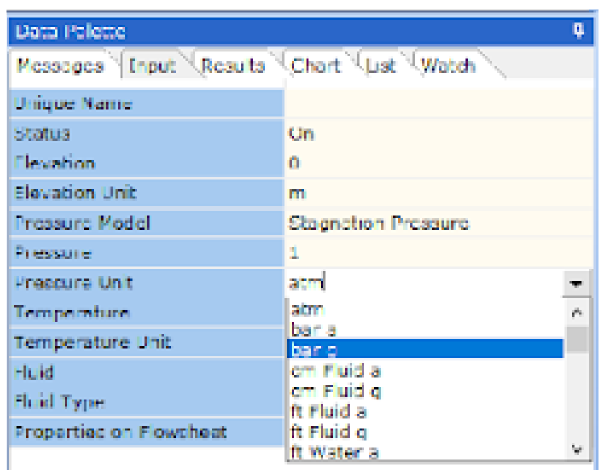


Figure 9: Pressure Units.

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

In this cell…	Enter…
Elevation	20
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):

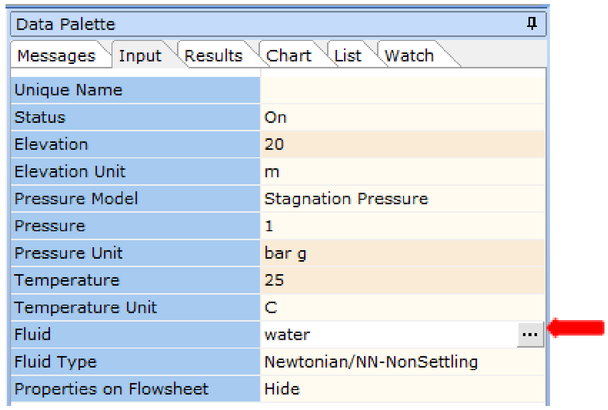


Figure 10: Fluid with 3 dots button.

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

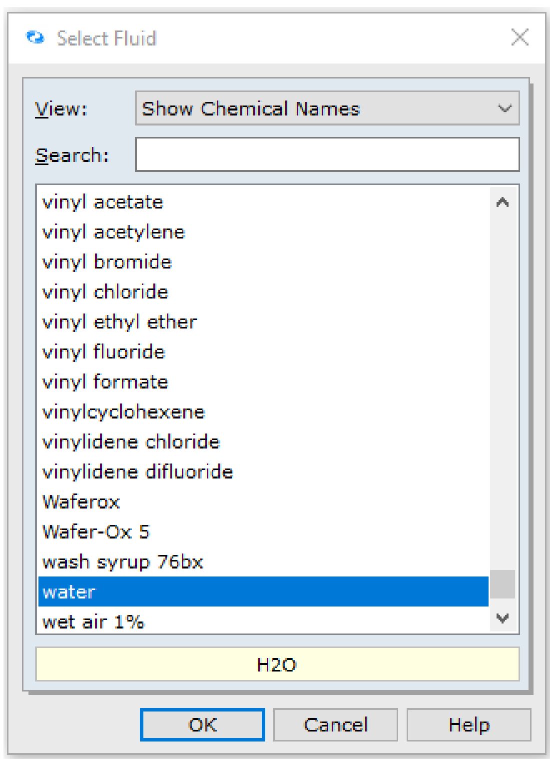


Figure 11: Select Fluid Dialog Box.

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).

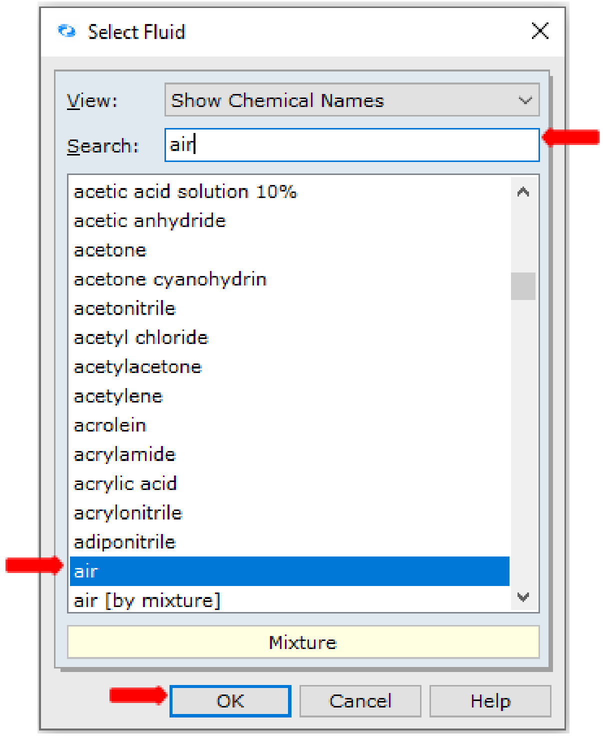


Figure 12: Select Fluid Dialog Box.

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

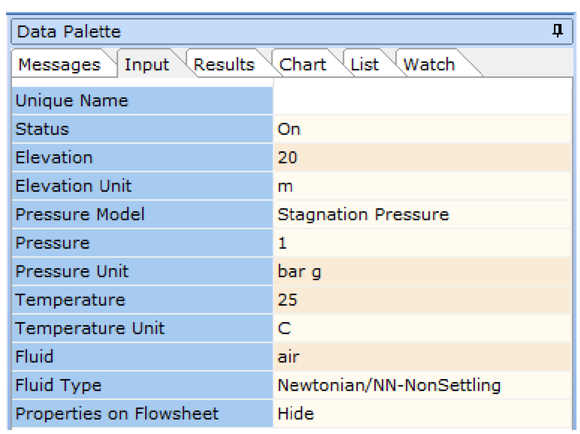
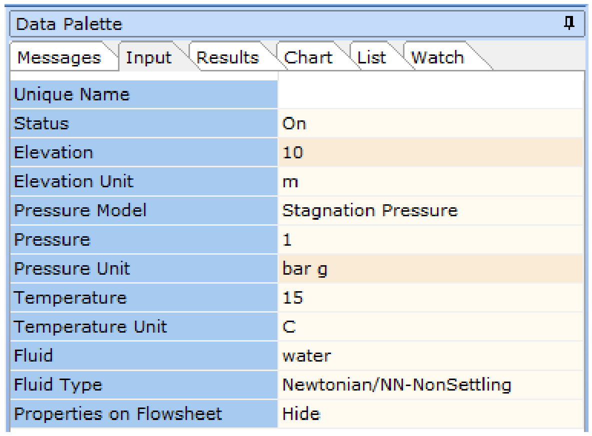


Figure 13: Input Inspector at Node 1.

12. We can now change the outlet elevation by selecting node 2. View the *Data Palette* and click the *Input Tab*. Enter the specification:

In this cell…	Enter…
Elevation	10

The Input Inspector on the *Data Palette* should now look like the screenshot given below (Figure 14).


Data Palette

Messages | Input | Results | Chart | List | Watch

Unique Name	
Status	On
Elevation	10
Elevation Unit	m
Pressure Model	Stagnation Pressure
Pressure	1
Pressure Unit	bar g
Temperature	15
Temperature Unit	C
Fluid	water
Fluid Type	Newtonian/NN-NonSettling
Properties on Flowsheet	Hide

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 25ºC and air, respectively. Note that FluidFlow will use the operating conditions of the higher elevation 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 15).

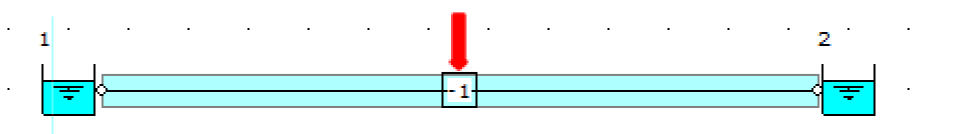


Figure 15: Highlighted Pipe.

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 16):

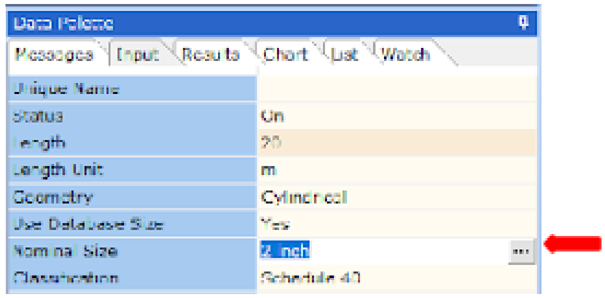


Figure 16: Nominal Size with 3 dots button.

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

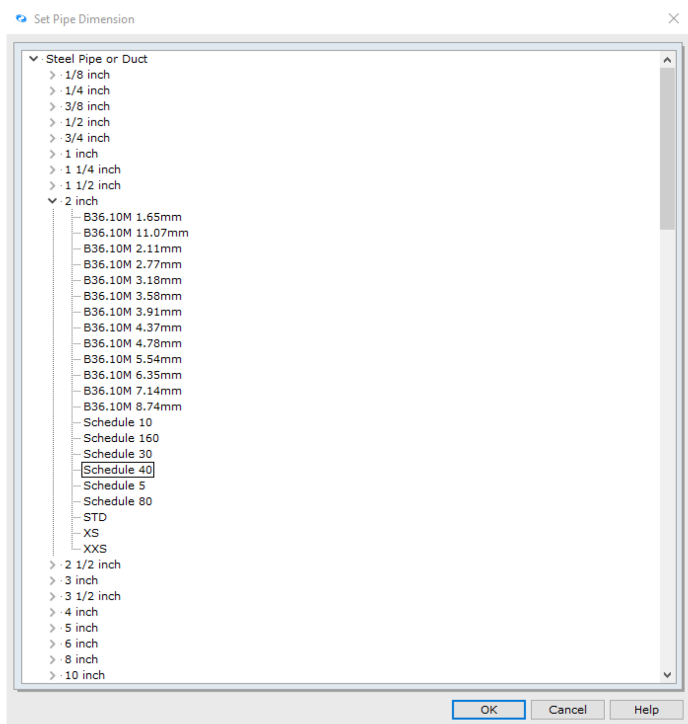


Figure 17: Set Pipe Dimension Dialog Box.

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).

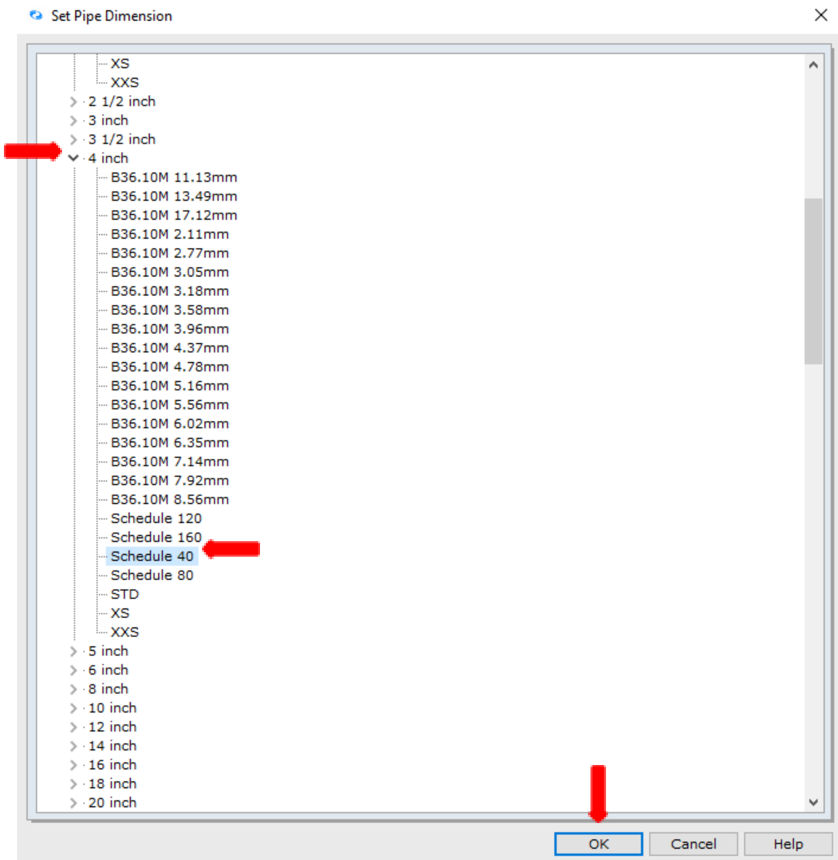


Figure 18: Set Pipe Dimension Dialog Box.

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

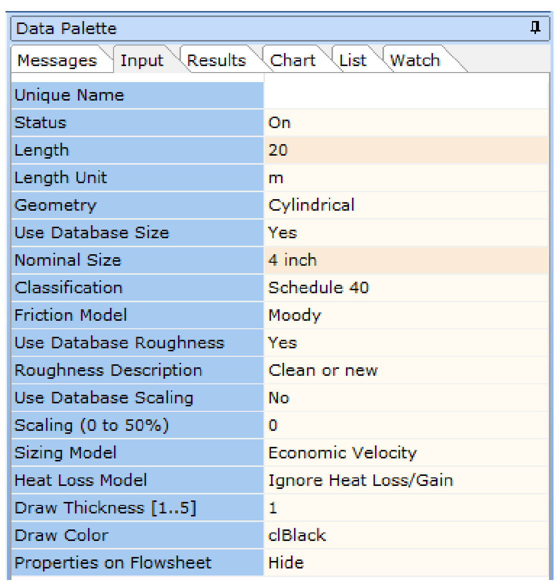


Figure 19: Input Inspector at Pipe Number “-1”.

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):

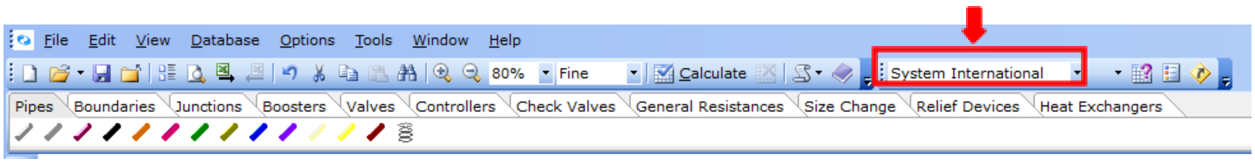


Figure 20: Unit Selector.

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).

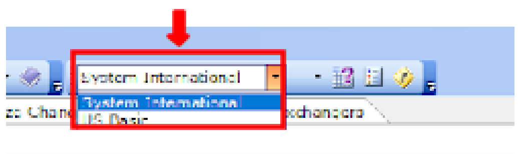


Figure 21: Unit Selector Drop-Down Menu.

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).

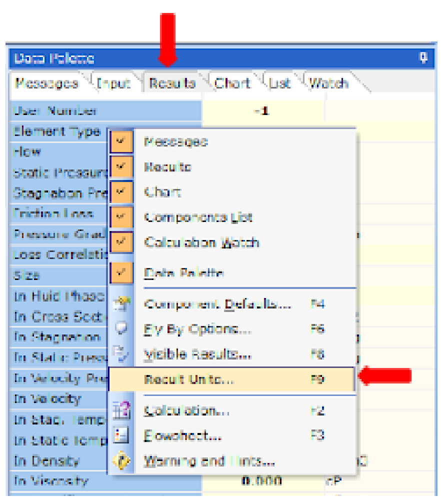


Figure 22: Result Units.

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

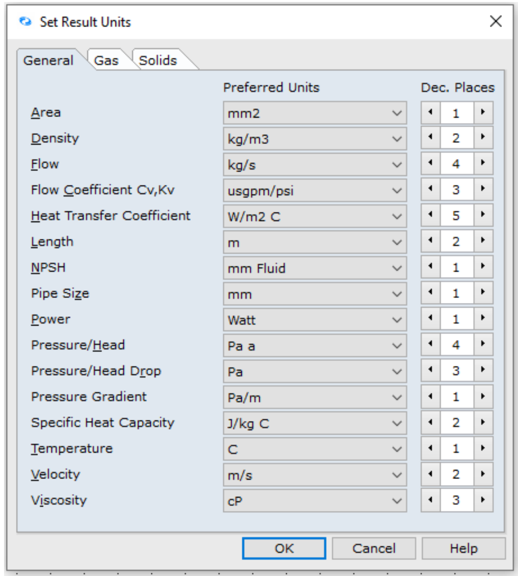


Figure 23: Set Result Units Dialogue Box.

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 dialogue box should now look like the screenshot given below (Figure 24). Select **OK** to apply the changes.

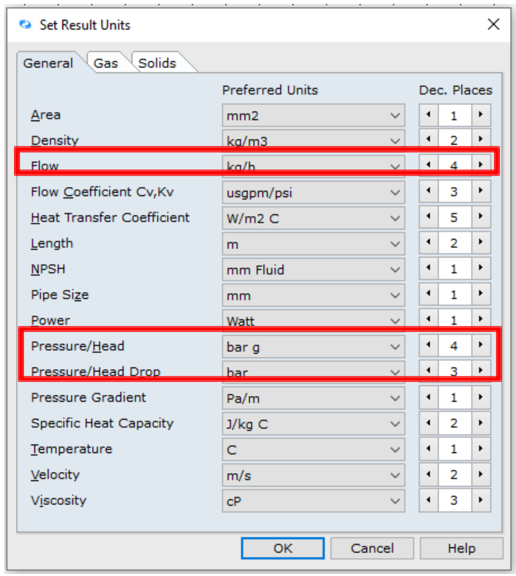


Figure 24: Modified Set Result Units Dialogue Box.

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).

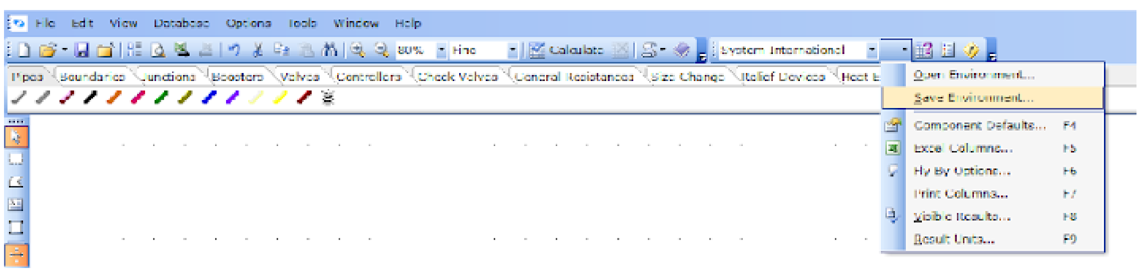


Figure 25: Save Environment Option.

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

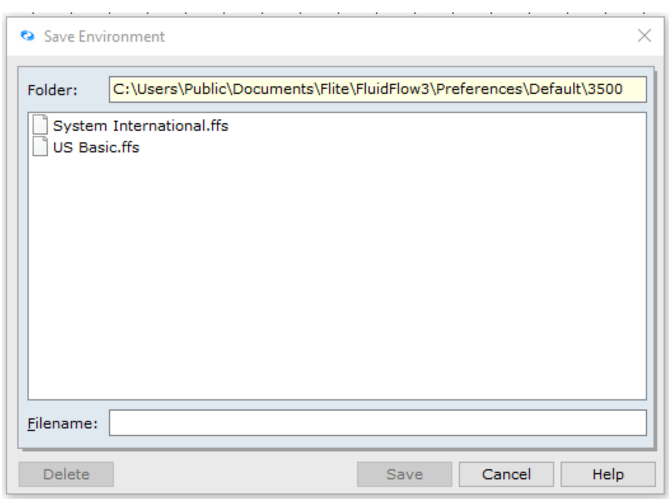


Figure 26: Save Environment Dialogue Box.

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

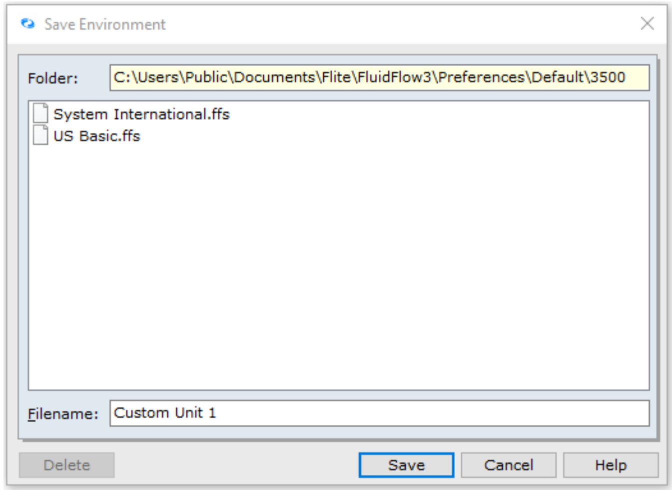


Figure 27: Modified Save Environment Dialogue Box.

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.

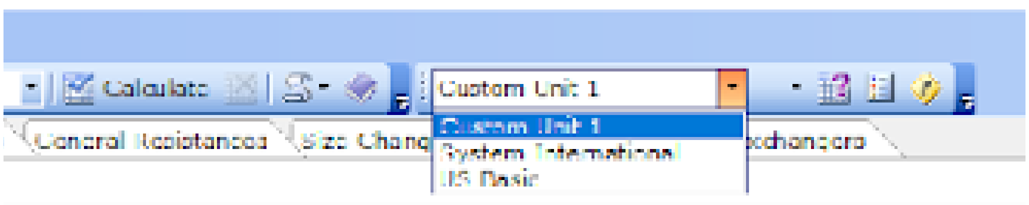


Figure 28: Unit Set Drop-Down Menu.

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):

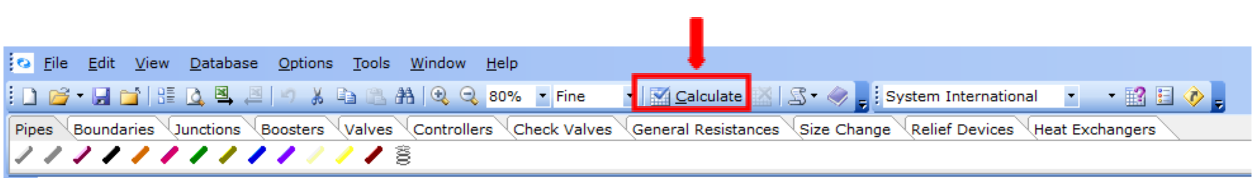


Figure 29: Calculate Button.

30. Check the converged hydraulic model in the flowsheet. The blue arrowhead shows that the flow direction is from the higher elevation node 1 to the lower elevation 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).

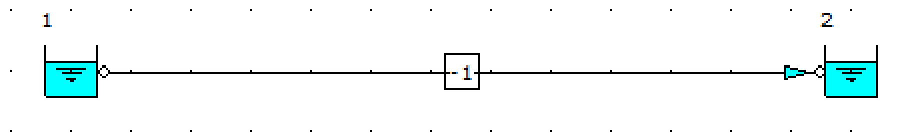


Figure 30: Converged Hydraulic Model

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

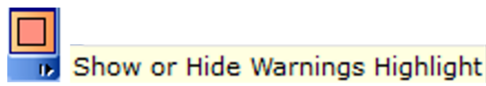


Figure 31: Show or Hide Warnings Highlight Button.

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.

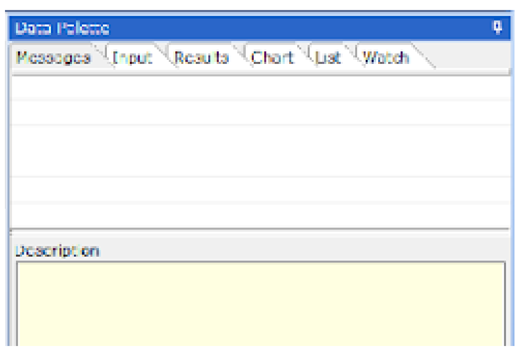


Figure 32: Warning Messages Tab.

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 | Chart | List | Watch

User Number	-1	
Element Type	Steel Pipe, Duct or Tube	
Flow	485.3885	kg/h
Flow at STP	396.0	m3/h
Flow at NTP	375.3	m3/h
Static Pressure Loss	0.000	bar
Stagnation Pressure Loss	0.000	bar
Friction Loss	0.002	bar
Pressure Gradient	11.5	Pa/m
Loss Correlation	Duxbury	
Economic Velocity	9.01	m/s
Exact Economic Size	89.9	mm
Size	102.3	mm
In Fluid Phase	Gas or Vapor	
In Cross Section Flow Area	8219.4	mm2
In Stagnation Pressure	1.0000	bar g
In Static Pressure	0.9994	bar g
In Velocity Pressure	0.001	bar
In Velocity	6.97	m/s
In Mach Number	0.02	
In Stag. Temperature	25.0	C
In Static Temperature	25.0	C
In Density	2.35	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	0.9994	bar g
Out Velocity Pressure	0.001	bar
Out Velocity	6.97	m/s
Out Mach Number	0.02	
Out Stag. Temperature	25.0	C
Out Static Temperature	25.0	C
Out Density	2.35	kg/m3
Out Viscosity	0.018	cP
Out Specific Heat Capacity	1007.47	J/kg C
Composition Mass %	air	100.0%
Reynolds No	91349.0	
Friction Factor	0.020530	

- 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).

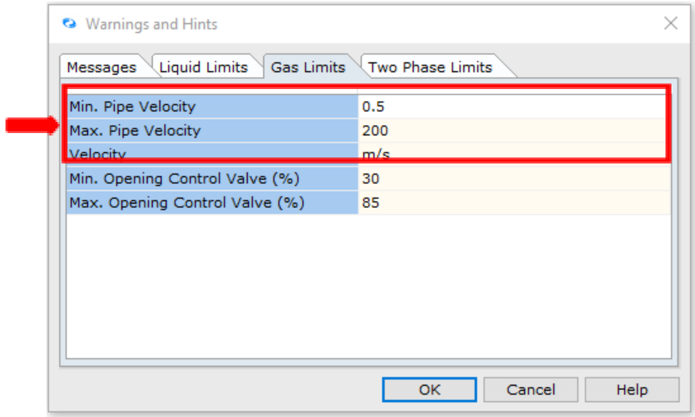


Figure 34: Warning and Hints: Gas Limits.

34. As we can see in Figure 33, the actual flowing velocity of the fluid is **6.97 m/s** 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): 1.00 barg.
    Stagnation Pressure at outlet boundary (Node 2): 1.00 barg.
    Stagnation Pressure Loss: 0.000 bar.

    We know this is correct since we had an inlet pressure of **1.00 barg** and outlet pressure of **1.00 barg** which gives us a difference of **0.000 bar** (Figure 33). Since the stagnation pressure for inlet and outlet pressure points are both 1 bar gauge, we cannot rely on the pressure difference as a driving force for the gas to flow. Therefore, we will rely on the 10 meters elevation difference between the inlet and outlet boundaries, to drive the gas to flow via gravity.

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

Parameter	Value
Flowrate	485 kg/h
Reynolds Number	91349
Darcy Friction Factor	0.02053
Friction loss	0.002 bar

Note that for gas flow applications, changes in elevation are negligible. The density of gases is relatively low such that the static head term, of the mechanical energy balance equation for a gas system, does not significantly contribute. This is evident in the small value of the calculated flow rate, and frictional loss, across the 20-m pipe length, in our example.

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).

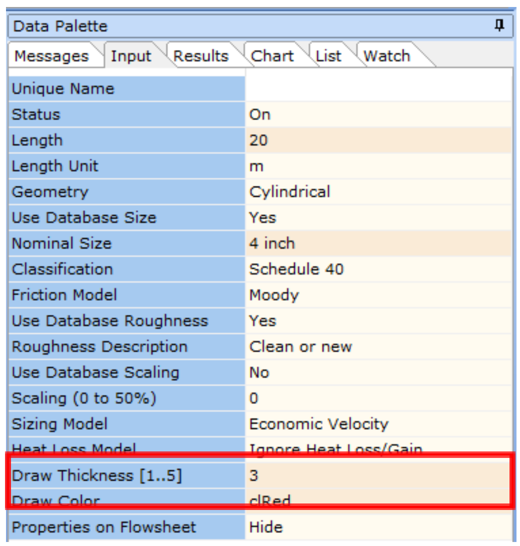


Figure 35: Modified Draw Thickness and Draw Color.

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

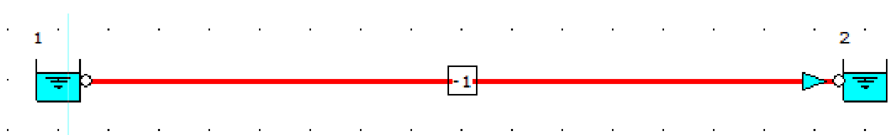


Figure 36: Hydraulic Model with Modified Draw Thickness and Draw Color.

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).

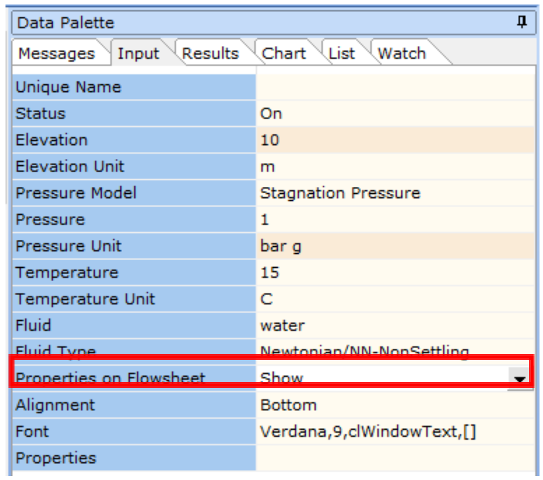


Figure 37: Show Properties on Flowsheet.

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

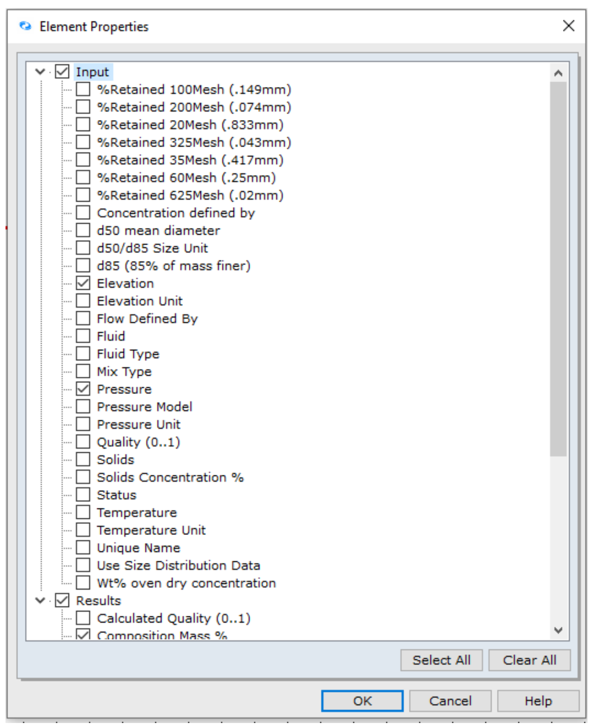


Figure 38: Element Properties Dialogue Box.

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

Element Properties	
Input	Results
Elevation Pressure	Composition Mass % Density Temperature Viscosity

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

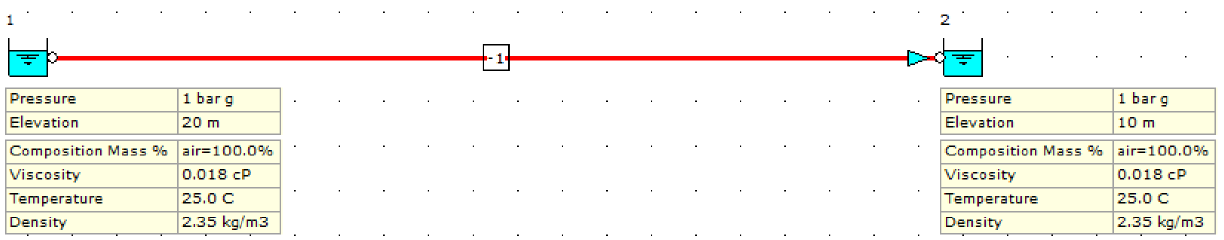


Figure 39: Hydraulic Model with Inlet and Outlet Boundary Properties.

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).

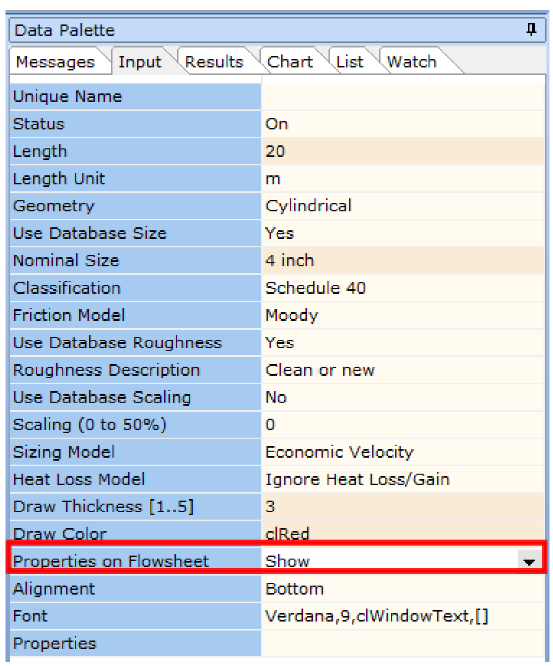


Figure 40: Show Properties on Flowsheet.

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

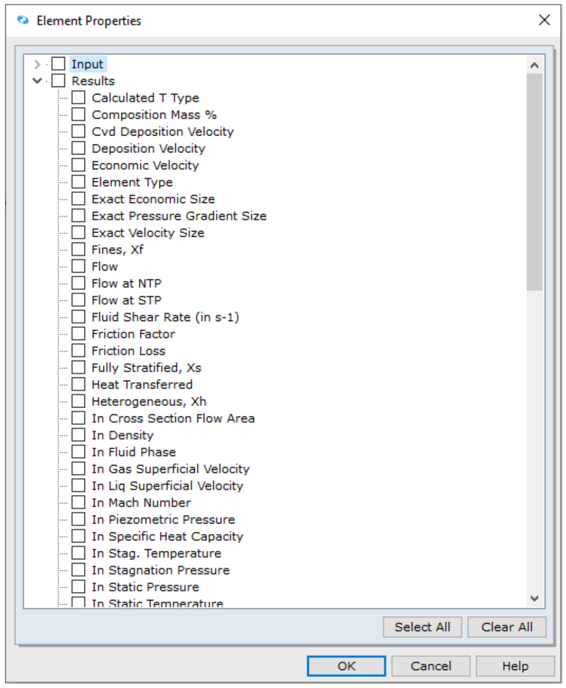


Figure 41: Element Properties Dialogue Box.

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

Element Properties	
Input	Results
Length	Flow
Nominal Size	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).

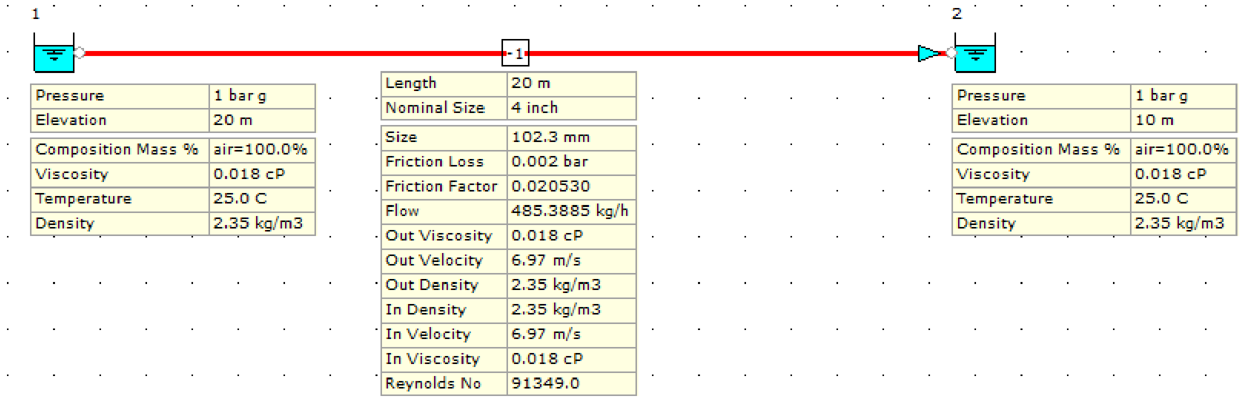


Figure 42: Hydraulic Model with Pipe Properties.

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.

$Given:$

$P_1 = 1.0\ barg$

$P_2 = 1.0\ barg$

$z_1 = 20\ m$

$z_2 = 10\ m$

$4"\ SCH\ 40$

$L = 20.0\ m$

$Air\ at\ T = 25°C = 298.15\ K$

$Required:$

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

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

$Velocity,\ V\ =\ ?$

$Reynolds\ Number,\ Re\ =\ ?$

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

$Friction\ loss\ =\ ?$

$Solution:$

$z_1\ =\ 20\ m$

$z_2\ =\ 10\ m$

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

$Density\ of\ Air\ at\ 25.0°C,\ 1.1\ barg,\ 1.0\ barg\ from\ FluidFlow\ Calculated\ Properties\ Database$

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

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

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

$For\ compressible\ fluids\ in\ which\ the\ pressure\ drop\ is\ relatively\ low\ and\ the\ density\ and\ velocity\ do\ not\ change\ appreciably,\ the\ flow\ may\ be\ treated\ as\ incompressible.\ Thus,\ equations\ used\ in\ incompressible\ calculations\ can\ be\ applied\ in\ our\ example.$

$Mechanical\ Energy\ Balance\ Equation:$

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

$Summation\ of\ Frictional\ Losses:$

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

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

$W'\ =\ 0\ (No\ compressor)$

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

$\frac{\Delta P}{\rho} = 0 \left(Constant\ pressure, P_1 = P_2\right)$

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

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

Applying the above conditions to Mechanical Energy Balance equation:

$$0 = \Delta z \frac{g}{g_c} + 0 + 0 + \frac{4f_F LV^2}{2Dg_c}$$

$$0 = \left(z_2 - z_1\right)\frac{g}{g_c} + \frac{4f_F LV^2}{2Dg_c}$$

$$\left(z_1 - z_2\right)\frac{g}{g_c} = \frac{2f_F LV^2}{Dg_c}$$

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

$$f_F V^2 = \frac{D\left(z_1 - z_2\right)g}{2L} = \frac{(0.1023\ m)(20-10)\ m\left[9.80665 \frac{m}{s^2}\right]}{2(20.0\ m)}$$

$$f_F V^2 = 0.2508050738 \rightarrow Equation\ 1$$

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

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

$$Re = \frac{(0.1023\ m)(V)\left(2.35 \frac{kg}{m^3}\right)}{0.018\ cP \times \frac{Pa-s}{1000\ cP}}$$

$$Re = \frac{80135}{6} V \rightarrow Equation\ 2$$

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

The Colebrook formula Equation (6 − 38)

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

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

Table 6 − 1 Value of Surface Roughness for Various Materials

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

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

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

$Trial\ and\ Error\ (Iteration)$

$Trial\ 1$

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

$using\ Equation\ 1.$

$f_F V^2 = 0.2508050738\ \rightarrow Equation\ 1$

$Assume\ f_F = 0.005$

$$V = \sqrt{\frac{0.25080507385}{0.005}}$$

$V = 7.0824\ \frac{m}{s}$

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

$Re = \frac{80135}{6} V \rightarrow Equation\ 2$

$Re = \frac{80135}{6}\left(7.0824\ \frac{m}{s}\right)$

$Re = 94591.354$

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

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

$$\frac{1}{\sqrt{f_F}} = -\ 4 \log log \left[\frac{0.0457\ mm}{3.7(102.3\ mm)} + \frac{1.256}{94591.354\sqrt{f_F}}\right]$$

$f_F = 5.0659\times10^{-3}$

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

$$0.005 \neq 5.0659\times10^{-3}$$

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

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

$Trial\ 2:$

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

$using\ Equation\ 1.$

$$f_F V^2 = 0.2508050738 \rightarrow Equation\ 1$$

$$Assume\ f_F = 5.0659\times10^{-3}$$

$$V = \sqrt{\frac{0.2508050738}{5.0659\times10^{-3}}}$$

$$V = 7.0362\ \frac{m}{s}$$

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

$$Re = \frac{80135}{6}V \rightarrow Equation\ 2$$

$$Re = \frac{80135}{6}\left(7.0362\ \frac{m}{s}\right)$$

$$Re = 93974.3145$$

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

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

$$\frac{1}{\sqrt{f_F}} = -\ 4\log log\left[\frac{0.0457\ mm}{3.7(102.3\ mm)} + \frac{1.256}{93974.3145\sqrt{f_F}}\right]$$

$$f_F = 5.0706\times10^{-3}$$

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

$$5.0659\times10^{-3} \neq 5.0706\times10^{-3}$$

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

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

$Trial\ 3$:

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

$using\ Equation\ 1$.

$f_F V^2 = 0.2508050738 \rightarrow Equation\ 1$

$Assume\ f_F = 5.0706\times10^{-3}$

$V = \sqrt{\frac{0.2508050738}{5.0706\times10^{-3}}}$

$V = 7.0330\ \frac{m}{s}$

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

$Re = \frac{80135}{6} V \rightarrow Equation\ 2$

$Re = \frac{80135}{6}\left(7.0330 \frac{m}{s}\right)$

$Re = 93931.5758$

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

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

$\frac{1}{\sqrt{f_F}} = -\ 4 \log log \left[\frac{0.0457\ mm}{3.7(102.3\ mm)} + \frac{1.256}{93931.5758\sqrt{f_F}}\right]$

$f_F = 5.0709\times10^{-3}$

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

$$5.0706\times10^{-3} \neq 5.0709\times10^{-3}$$

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

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

$Trial\ 4$:

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

$using\ Equation\ 1.$

$f_F V^2 = 0.2508050738 \rightarrow Equation\ 1$

$Assume\ f_F = 5.0709\times10^{-3}$

$V = \sqrt{\frac{0.2508050738}{5.0709\times10^{-3}}}$

$V = 7.0328\frac{m}{s}$

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

$Re = \frac{80135}{6}V \rightarrow Equation\ 2$

$Re = \frac{80135}{6}\left(7.0328\frac{m}{s}\right)$

$Re = 93928.9047$

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

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

$\frac{1}{\sqrt{f_F}} =- 4\log log\left[\frac{0.0457\ mm}{3.7(102.3\ mm)} + \frac{1.256}{93928.9047\sqrt{f_F}}\right]$

$f_F = 5.0709\times10^{-3}$

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

$$5.0709\times10^{-3} = 5.0709\times10^{-3}$$

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

$Stop\ the\ iteration.$

$Summary\ of\ Iteration$

$Trial$	$f_{F,\ assumed}$	$V_{calculated}$	$Re_{calculated}$	$f_{F,\ calculated}$
	0.005	7.0824	94591.3540	5.0659×10^{-3}
2	5.0659×10^{-3}	7.0362	93974.3145	5.0706×10^{-3}

3	5.0706×10^{-3}	7.0330	93931.5758	5.0709×10^{-3}
4	5.0709×10^{-3}	7.0328	93928.9047	5.0709×10^{-3}

$Therefore$

$f_F = 5.0709\times10^{-3}$

$V = 7.0328\frac{m}{s}$

$Re = 93928.9047$

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

$f_D = 4f_F = 4\left(5.0709\times10^{-3}\right)$

$f_D = 0.0203$

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

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

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

$Friction\ Loss = -\Delta P = \frac{2\left(5.0709\times10^{-3}\right)(20\ m)\left(7.0328\frac{m}{s}\right)^2\left(2.35\frac{kg}{m^3}\right)}{(0.1023\ m)\left(1\frac{kg-m}{N-s^2}\right)} = 230.4591\ Pa$

$Friction\ Loss = -\Delta P = 230.4591\ Pa\ Pa\times\frac{1.01325\ bar}{101325\ Pa}$

$Friction\ Loss = -\Delta P = 0.002\ bar$

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

$\dot{q} = VS = V\left(\frac{\pi}{4}D^2\right) = 7.0328\frac{m}{s}\left[\left(\frac{\pi}{4}\right)(0.1023\ m)^2\right]\left[\frac{3600\ s}{1\ h}\right]$

$\dot{q} = 208.0999\frac{m^3}{h}$ $Calculate\ for\ the\ value\ of\ volumetric\ flow\ rate,\ \dot{q}$

$\dot{m} = \rho_{Air}VS = \rho_{Air}V\left(\frac{\pi}{4}D^2\right) = \left(2.35\frac{kg}{m^3}\right)\left(7.0328\frac{m}{s}\right)\left[\left(\frac{\pi}{4}\right)(0.1023\ m)^2\right]\left[\frac{3600\ s}{1\ h}\right]$

$\dot{m} = 489.0348\frac{kg}{h}$

Results Comparison:

Description	Unit	FluidFlow Results	Hand Calculation	% Difference
	kg/h	485	489	0.8214%
Volumetric Flow Rate, q	m^3/h	206	208	0.9662%
Velocity, V	m/s	6.97	7.03	0.8571%
Reynolds Number, Re	-	91349	93929	2.7850%
Darcy Friction Factor, f	-	0.0205	0.0203	0.9804%
Friction loss	Bar	0.002	0.002	0%

Commentary:

The results compare very well with the hand calculation.