FluidFlow

The Flow of Liquid via Pressure Difference

© Flite Software NI Ltd





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 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 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).

	85 0																		Syst							3 !	9	=																					
	: Qur					introl	lers	Che	eck Vi	alves	Ge	enera	Resi	stance	s \s	ize Cł	nange	Re	lief D	evice	ss V	feat l	Excha	inger	s \																								
-	-	-	-	 <u> </u>	8																			_					 _	 	_	 	 	 	 	 		0.0	Oata	Palet		_	_	_	_	_	-	-	-
																																					- 1				a Ing	put	Rep	sults	Ch	art	List	Wa	ch

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).

Pip	es Boundaries Junctions Boosters
⊡	🚈 🛱 नी 🦡 🖷 🧕
13	Known or Assigned Pressure
\square	

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).



1 .	•	•	•	•	•	•	•	•	•	•	2 '
Ŧ											Ŧ

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.

 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):



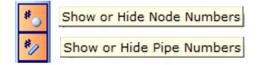


Figure 6: Show or Hide Node/Pipe Numbers buttons.

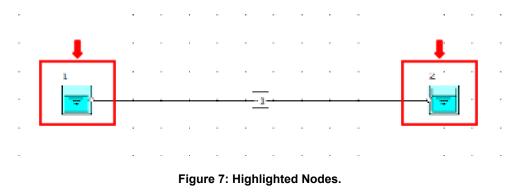
The automatic assignment of *User Numbers* helps us differentiate each of the nodes when we examine our systems – post calculation. This will become more apparent later.

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).



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



Data Palette 👆	Ф
Messages Input Results	Chart List Watch
Unique Name	
Status	On
Elevation	0
Elevation Unit	m
Pressure Model	Stagnation Pressure
Pressure	1
Pressure Unit	atm
Temperature	15
Temperature Unit	с
Fluid	water
Fluid Type	Newtonian/NN-NonSettling
Properties on Flowsheet	Hide

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.

Data Polette		Ģ
Mossoges Input Results	Chart Ust Watch	
Unique Name		
Status	Un	
Clevation	0	
Elevation Unit	m	
Pressure Model	Stagnation Pressure	
Pressure	1	
Pressure Unit	atm	-
Temperature	atm	^
Temperature Unit	bar a bar o	
Huid	cm Fluid a	
Fluid Type	om Fluid g ft Fluid a	
Properties on Flowsheat	ft Fluid q	
	ft Water a	~

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).



Data Polette	Ģ							
Messages Input Reaults Chart Ust Watch								
Unique Name								
Status	Un							
Clevation	0							
Elevation Unit	m							
Pressure Model	Stagnation Pressure							
Pressure	1.1							
Pressure Unit	bar q							
Temperature	25							
Temperature Unit	c							
Huid	wator							
Fluid Type	Newtonian/NN-NonSettling							
Properties on Flowsheet	Hide							

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).

Data Polette	9							
Messages Input Reaults Chart List Watch								
Unique Name								
Status	On							
Elevation	0							
Elevation Unit	m							
Pressure Model	Stagnation Preasure							
Pressure	1							
Pressure Unit	bar q							
Temperature	15							
Temperature Unit	c							
Huid	water							
Fluid Type	Newtonian/NN-NonSettling							
Properties on Flowsheet	Hide							

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 25°C. 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).

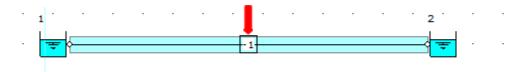


Figure 12: Highlighted Pipe.



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):

Data Polette	Ģ
Messages Input Results	Chort Ust Watch
Unique Name	
Status	On
Length	20
Length Unit	m
Geometry	Cylindrical
Use Database Size	Yes
Nominal Size	2 inch
Classification	Schedule 40

Figure 13: Nominal Size with 3 dots button.

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

Steel Pine of Duct	
Steel Pipe or Duct 1/8 inch	
> 1/4 inch	
> 3/8 inch	
> 1/2 inch	
> 3/4 inch	
> 1 inch	
> 1 1/4 inch	
> 1 1/2 inch	
✓ 2 inch	
B36.10M 1.65mm	
- B36.10M 11.07mm	
- B36.10M 2.11mm	
B36.10M 2.77mm	
B36.10M 3.18mm	
B36.10M 3.58mm	
- B36.10M 3.91mm	
- B36.10M 4.37mm	
B36.10M 4.78mm	
B36.10M 5.54mm	
B36.10M 6.35mm	
- B36.10M 7.14mm	
B36.10M 8.74mm	
- Schedule 10	
···· Schedule 160	
Schedule 30	
Schedule 40	
Schedule 5	
Schedule 80	
···· STD	
···· XS	
XXS	
> 2 1/2 inch	
> · 3 inch	
> 3 1/2 inch	
> 4 inch	
> 5 inch	
> 6 inch	
> 8 inch	
> 10 inch	

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).

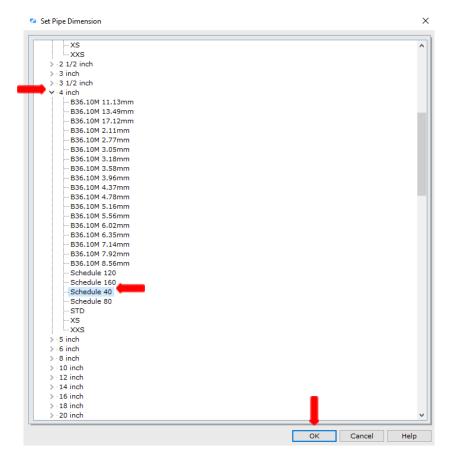


Figure 15: Set Pipe Dimension Dialog Box.

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



Data Palette	д
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 [15]	1
Draw Color	clBlack
Properties on Flowsheet	Hide

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):

<u> </u>	<u>E</u> dit	<u>V</u> iew	<u>D</u> atabase	<u>O</u> ptions	Tools	<u>W</u> indow	<u>H</u> elp				-
i 🗅 💕	- 🗐 (1 85	👌 📓 🛤	9 % [b (b	船 🔍 🤤	80%	 Fine 	🔹 🗹 Calculate 🔣 🖾 🔹 🧼	System International	🚽 🕘 🔛 🔶 💂
Pipes	Bounda	ries (Junctions B	oosters	Valves	Controlle	rs (C	heck Valves	General Resistances Size Cl	nange Relief Devices Hei	at Exchangers
11	11	11	111	11.	///	ê					

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).

Data Polette	۹
Messages (Input	Results Chort List Watch
User Number	-1
Element Type	Messages
Static Pressure	Recuito
Stagnation Pre 🗹	Chart
Friction Loss 🗸	Components List
Pressure Grad	Calculation Watch
5 ze	Data Palatta
In Huid Phase	Component <u>D</u> efaults 74
In Stacnation	Ely By Options P6
In Static Press 💱	Visible Results F8
In Velocity Pre	Recuit Units P9
In Velocity	Calculation F2
In Static Temp	Elowsheet F3
In Density 🛛 🤣	Werning and Lints 0
In Viscosity	0.000 cP

Figure 19: Result Units.

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

Set Result Units ×							
General Gas Solids							
	Preferred Units						
<u>A</u> rea	mm2	~	•	1	۲		
<u>D</u> ensity	kg/m3	~	4	2	۲		
<u>F</u> low	kg/s	~	4	4	٠		
Flow Coefficient Cv,Kv	usgpm/psi	~	•	3	۲		
Heat Transfer Coefficient	W/m2 C	~	•	5	•		
<u>L</u> ength	m	\sim	•	2	•		
NPSH	mm Fluid	\sim	•	1	•		
Pipe Size	mm	\sim	•	1	•		
<u>P</u> ower	Watt	~	•	1	•		
Pressure/ <u>H</u> ead	Pa a	~	4	4	۲		
Pressure/Head Drop	Pa	~	4	3	٠		
Pressure Gradient	Pa/m	~	•	1	۲		
Specific Heat Capacity	J/kg C	~	•	2	۰		
Temperature	С	~	•	1	·		
Velocity	m/s	\sim	•	2	•		
Viscosity	cP	\sim	•	3	·		
	ОК	Cancel		Hel	p		

Figure 20: Set Result Units Dialogue Box.

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

In this unit	Select
Flow	m³/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.

(-
	Preferred Units		De	c. Pla	ice
<u>A</u> rea	mm2	\sim	•	1	•
<u>D</u> ensity	kg/m3	~	4	2	•
Flow	m3/h	\sim	4	4	•
Flow Coefficient Cv,Kv	usgpm/psi	\sim	•	3	•
Heat Transfer Coefficient	W/m2 C	\sim	•	5	•
Length	m	\sim	•	2	•
NPSH	mm Fluid	~	4	1	•
Pipe Size	mm	~	4	1	•
<u>P</u> ower	Watt	~	4	1	•
Pressure/ <u>H</u> ead	bar g	~	4	4	•
Pressure/Head Drop	bar	~	4	3	•
Pressure Gradient	Pa/m	~	4	1	•
Specific Heat Capacity	J/kg C	~	•	2	,
Temperature	С	~	1	1	•
<u>V</u> elocity	m/s	~	•	2	•
Viscosity	cP	~	•	3	

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).



Save Envi	ironment			×
Folder:	C:\Users\Public\Documents	\Flite\FluidFlow3\Pre	eferences\Defa	ault\3500
	10-113			
<u>F</u> ilename:				
Delete		Save	Cancel	Help

Figure 23: Save Environment Dialogue Box.

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

Save Env	ironment	\times
Folder:	C:\Users\Public\Documents\Flite\FluidFlow3\Preferences\Default\3500	
US Bas	n International.ffs ,ic.ffs	
<u>F</u> ilename:	Custom Unit 1	
Delete	Save Cancel Help	

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.

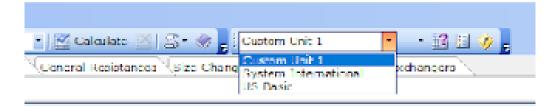


Figure 25: Unit Set Drop-Down Menu.

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):

🐼 Eile Edit View Database Options Tools Window Help
: 🗋 🚰 マ 🚛 🖆) 語 🐧 🚇 🖉 🔌 👒 🛝 🐴 🔍 🔍 80% 🔹 Fine 🛛 💌 Calculate 🔣 🖾 🗞 System International 🔹 🔹 🔢 🚱 📮
Pipes Boundaries Junctions Boosters Valves Controllers Check Valves General Resistances Size Change Relief Devices Heat Exchangers

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).

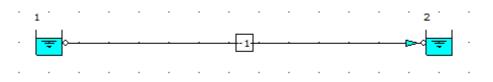


Figure 27: Converged Hydraulic Model

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

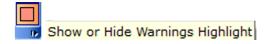


Figure 28: Show or Hide Warnings Highlight Button.

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.

Data Polette	ų
Messages (Input (Results (Chart (List (Watch)	
Icacription	

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		д	1
	Messages Input Results	Chart List W	'atch	
	User Number	-1]
	Element Type	Steel Pipe, Duc	t 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	- Suggested Economic Pipe Ø.
	Size	102.3	mm	- Actual I.D. of 4" Sch. 40 Pipe.
<u> </u>	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	
The "In" values	In Velocity Pressure	0.028	bar	
represent the calculated —	In Velocity	2.37	m/s	
conditions at the	In Stag. Temperature	25.0	с	
pipe Inlet.	In Static Temperature	25.0	С	
	In Density	997.10	kg/m3	
	In Viscosity	0.890	сP	
	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	
The "Out" values	Out Velocity Pressure	0.028	bar	
represent the calculated — conditions at the	Out Velocity	2.37	m/s	
pipe Outlet.	Out Stag. Temperature	25.0	С	
pipe outlet.	Out Static Temperature	25.0	С	
	Out Density	997.09	kg/m3	
	Out Viscosity	0.890	сP	
	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		

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.



Since we are modeling a liquid system, we are only interested in the Liquid Limits (Figure 31).



Min. Pipe Velocity		0.2		
Max. Pipe Velocity		4		
Velocity		m/s		
Min. Opening Control Val	ve (%)	20		
Max. Opening Control Va	lve (%)	75		

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 m³/h					
Reynolds Number	271491					
Darcy Friction Factor	0.018282					
Friction loss	0.100 bar					



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).

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 [15]	3							
Draw Color	clRed							
Properties on Flowsheet	Hide							

Figure 32: Modified Draw Thickness and Draw Color.

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

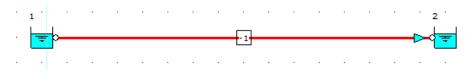


Figure 33: Hydraulic Model with Modified Draw Thickness and Draw Color.

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).

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	с								
Fluid	water								
Eluid Type	Newtonian/NN-NonSettling								
Properties on Flowsheet	Show								
Alignment	Bottom								
Font	Verdana,9,clWindowText,[]								
Properties									



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):

Element Properties			>
V. Input			^
Retained 100Mesh (.149mm)			
%Retained 20Mesh (.833mm)			
%Retained 325Mesh (.043mm)			
Retained 35Mesh (.417mm)			
Retained 60Mesh (.25mm)			
Retained 625Mesh (.02mm)			
Concentration defined by			
d50 mean diameter			
d50/d85 Size Unit			
d85 (85% of mass finer)			
✓ Elevation			
Elevation Unit			
Flow Defined By			
Fluid			
Fluid Type			
Pressure			
Pressure Model			
Pressure Unit			
Quality (01)			
Solids			
Solids Concentration %			
Status			
Temperature			
Temperature Unit			
Unique Name			
Use Size Distribution Data			
Wt% oven dry concentration			
V Results			
Calculated Quality (01)			
Composition Mass %			~
		Select All	Clear All
	ОК	Cancel	Help

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	Density						
Fluid	Temperature						
Pressure	Viscosity						

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



1															2	
 =						- 1								\triangleright	d ⊊ ·	
Fluid	water].													Fluid	water
Pressure	1.1 barg	1													Pressure	1 barg
Elevation	0 m	·													Elevation	0 m
Viscosity	0.890 cP]													Viscosity	0.890 cP
Temperature	25.0 C	1 ·	•	•	•		•	•	•	•		•	•	•	Temperature	25.0 C
Density	997.10 kg/m3	1													Density	997.09 kg/m3

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).

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 [15]	3							
Draw Color	clRed							
Properties on Flowsheet	Show 👻							
Alignment	Bottom							
Font	Verdana,9,clWindowText,[]							
Properties								

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):



Element Properties			×
> Input			^
✓ Results			
Calculated T Type			
Composition Mass %			
Cvd Deposition Velocity			
Deposition Velocity			
- Economic Velocity			
- Element Type			
Exact Economic Size			
Exact Pressure Gradient Size			
Exact Velocity Size			
Fines, Xf			
Flow			
Flow at NTP			
Flow at STP			
Fluid Shear Rate (in s-1)			
Friction Factor			
Friction Loss			
Fully Stratified, Xs			
Heat Transferred			
Heterogeneous, Xh			
In Cross Section Flow Area			
In Density			
In Fluid Phase			
In Gas Superficial Velocity			
In Lig Superficial Velocity			
In Mach Number			
In Piezometric Pressure			
In Specific Heat Capacity			
In Stag. Temperature			
In Stagnation Pressure			
In Static Pressure			
In Static Temperature			~
	[Select All	Clear All
Г	ОК	Cancel	Help

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								



1									•	•	•		·		2	•	•	•	•
 _					[- 1									÷				
Fluid	water].			Length	20 m									Fluid		water		
Pressure	1.1 barg				Nominal Size	4 inch									Pressu	ire	1 bar	g	
Elevation	0 m] .	•	• •	Size Friction Loss	102.3 mm 0.100 bar	•	•	•	•		•	·	·	Elevat	ion	0 m		
Viscosity	0.890 cP 25.0 C				Friction Factor	0.018282									Viscos	· ·	0.890		
Temperature Density	25.0 C 997.10 kg/m3				Flow	70.0975 m3/h									Densit		997.0		/m3
		J .	•	• •	Out Viscosity Out Velocity	0.890 cP 2.37 m/s	•	•	•	•	•	•	·	•			L		
					Out Density	997.09 kg/m3													
					In Density	997.10 kg/m3													
• •				• •	In Velocity	2.37 m/s	•				•	•	·	·		•		•	•
					In Viscosity Reynolds No	0.890 cP 271490.7													

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:

 $P_{1} = 1.1 \text{ barg}$ $P_{2} = 1.0 \text{ barg}$ 4" SCH 40 L = 20.0 mWater 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

$$\rho_{H_20} = 997.10 \, \frac{kg}{m^3}$$

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

$$\mu_{H_20} = 0.890 \, cP$$

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:

$$\begin{split} W &= 0 \ (No \ pump) \\ \Delta z \frac{g}{g_c} &= 0 \ (Horizontal \ Pipe, z_1 = z_2) \\ \frac{\Delta V^2}{2g_c} &= 0 \ (Constant \ cross \ - \ sectional \ area, \ V_1 = V_2) \\ F_{Fittings} &= F_{Contraction} = F_{Expansion} = F_{Metering} = 0 \\ \Sigma F &= F_{Line} = h_{Line} = \frac{4f_r L V^2}{2Dg_c} \end{split}$$

Applying the above conditions to Mechanical Energy Balance equation:

$$0 = 0 + 0 + \frac{\Delta P}{\rho} + \frac{4f_{F}LV^{2}}{2Dg_{c}}$$
$$0 = \frac{P_{2}-P_{1}}{\rho} + \frac{4f_{F}LV^{2}}{2Dg_{c}}$$



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

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

$$f_F V^2 = \frac{Dg_c (P_1 - P_2)}{2\rho L} = \frac{(0.1023 \, m) \left(1 \frac{kg - m}{N - s^2}\right) \left[(1.1 - 1) \, bar \times \frac{101325 \, Pa}{1.01325 \, bar}\right]}{2 \left(997.10 \frac{kg}{m^3}\right) (20.0 \, m)}$$
$$f_F V^2 = 0.02564938321 \rightarrow Equation 1$$

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 ϵ .

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

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 → Equation 2 $Re = 114610.4831(2.5323 \frac{m}{s})$ Re = 290228.1265

Step 3: Substitute the value of Re to Equation 3 and calculate the value of $f_{\rm F}$.

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

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

$$f_F = 4.4873 \times 10^{-3}$$

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

 $0.004 \neq 4.4873 \times 10^{-3}$ $f_{F, assumed} \neq f_{F, calculated}$

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 $f_F = 4.4873 \times 10^{-3}$



$$V = \sqrt{\frac{0.02564938321}{4.4873 \times 10^{-3}}}$$
$$V = 2.3908 \frac{m}{s}$$

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 $f_{\rm F}$.

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

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

$$f_F = 4.5079 \times 10^{-3}$$

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

4. 4873×10⁻³≠4. 5079×10⁻³
$$f_{F, assumed} \neq f_{F, calculated}$$

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
$$f_F = 4.5079 \times 10^{-3}$$

$$V = \sqrt{\frac{0.02564938321}{4.5079 \times 10^{-3}}}$$
$$V = 2.3853 \frac{m}{s}$$



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

$$Re = 114610.4831V → Equation 2$$

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

$$Re = 273380.3854$$

Step 3: Substitute the value of Re to Equation 3 and calculate the value of $f_{\rm F}$.

$$\frac{1}{\sqrt{f_F}} = -4 \log \log \left[\frac{0.0457 \text{ mm}}{3.7(102.3 \text{ mm})} + \frac{1.256}{\text{Re}\sqrt{f_F}} \right] \rightarrow Equation 3$$
$$\frac{1}{\sqrt{f_F}} = -4 \log \log \left[\frac{0.0457 \text{ mm}}{3.7(102.3 \text{ mm})} + \frac{1.256}{273380.3854\sqrt{f_F}} \right]$$
$$f_F = 4.5087 \times 10^{-3}$$

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

$$4.5079 \times 10^{-3} \neq 4.5087 \times 10^{-3}$$
$$f_{F, assumed} \neq f_{F, calculated}$$

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 $f_F = 4.5087 \times 10^{-3}$

$$V = \sqrt{\frac{0.02564938321}{4.5087 \times 10^{-3}}}$$
$$V = 2.3851 \frac{m}{s}$$

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 $f_{\rm F}$.

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

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

$$f_F = 4.5088 \times 10^{-3}$$

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

$$4.5087 \times 10^{-3} \neq 4.5088 \times 10^{-3}$$
$$f_{F, assumed} \neq f_{F, calculated}$$

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 $f_F = 4.5088 \times 10^{-3}$

$$V = \sqrt{\frac{0.02564938321}{4.5088 \times 10^{-3}}}$$
$$V = 2.3851 \frac{m}{s}$$

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 $f_{\rm F}$.

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

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

$$f_F = 4.5088 \times 10^{-3}$$

Step 4: Compare the assumed value of the $f_{_{\rm F}}$ to the calculated value.

4.
$$5088 \times 10^{-3} = 4.5088 \times 10^{-3}$$

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

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

 $f_F = 4.5088 \times 10^{-3}$

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

Re = 273357.4634

Calculate for the value of Darcy friction factor, $\boldsymbol{f}_{_{D}}$

$$f_D = 4f_F = 4(4.5088 \times 10^{-3})$$

 $f_D = 0.0180$



Calculate for the value of friction loss

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

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

Friction Loss =
$$\Delta P = \frac{2(4.5088 \times 10^{-3})(20 \text{ m})(2.3851\frac{m}{s})^2(997.10\frac{kg}{m^3})}{(0.1023 \text{ m})(1\frac{kg-m}{N-s^2})} = 10000 Pa$$

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

Friction Loss = $-\Delta P = 0.100 \text{ bar}$

Calculate for the value of volumetric flow rate, 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	m³/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.