FluidFlow

Equipment Sizing Handbook

QUICK GUIDE TO EQUIPMENT SIZING IN FLUIDFLOW

1. Introduction........ 2
2. Sizing Feature........ 2
3. Sizing Models........ 3
4. Pipe Sizing........ 4
4.1 Economic Velocity........ 4
4.2 By Velocity........ 6
4.3 By Pressure Gradient........ 7
4.4 Limitation of Pipe Sizing........ 8
5. Sample Equipment Sizing Models........ 8
5.1 Auto Size Pipes........ 8
5.2. Control Valve Sizing........ 14
5.3. Pump Sizing........ 18
5.4. Orifice Plate Sizing........ 21
5.5. Balancing Flow Distribution through Autosize........ 26

1. Introduction

FluidFlow software offers an easy and reliable approach on sizing pipes and associated equipment in transporting fluids. It significantly reduces engineering hours needed in designing or troubleshooting fluid systems without the worry of wrong calculation results typically encountered in hand or worksheet calculations.

"Sizing" referred to in this document, pertains to the determination of equipment characteristics needed to attain a specific design intent for a fluid transport system.

In the succeeding sections, the different aspects involved in sizing pipes and associated equipment in transporting fluids using FluidFlow will be discussed.

It is not the intention of this document to supersede any recognized standard nor applicable laws and regulations in designing systems. Should inconsistencies arise from this document, good engineering judgement should be applied in discerning what needs to be followed.

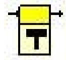
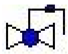
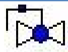
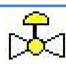
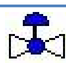
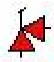
2. Sizing Feature

In FluidFlow, several components possess an input field to engage the Automatic sizing feature "ON" or "OFF".

When the automatic sizing feature is "ON", FluidFlow will perform a "sizing" calculation and determine component characteristics that will meet a design requirement.

When the automatic sizing feature is "OFF", FluidFlow will perform a "rating" calculation. A "rating" calculation means that the performance of a given component characteristic is calculated based on the conditions defined in the hydraulic model. This is usually achieved through the use of the database where most of the component characteristics for fluid transport are stored.

Table 2.1: Components with automatic sizing feature

Component Type	Component	Icon
Booster	Centrifugal Pump	
Booster	Rotating Positive Displacement Pump	
Booster	Fan or Compressor	
Booster	Positive Displacement Pump or Compressor	
Controller	Self Acting Pressure Reducer	
Controller	Self Acting Pressure Sustainer	
Controller	Self Acting Differential Pressure Controller	
Controller	Flow Control Valve	
Controller	Pressure Control Valve	
Size Change	Thin Orifice	
Size Change	Thick Orifice	
Size Change	Inline Nozzle	
Size Change	Venturi Tube	
Relief Device	Pressure Relief Valve	

3. Sizing Models

Sizing models in FluidFlow describe how the component characteristic will be determined by the solver.

As applicable, a component may have up to two sizing models to cater the typical input data available during equipment sizing.

Table 3.1: Component sizing models

Component Type	Sizing Model	"Sized" Component Characteristic*
Boosters	Size for Flow	Duty Pressure Rise
	Size for Pressure Rise	Duty Flow

Component Type	Sizing Model	"Sized" Component Characteristic*
Controllers	None	Flow Coefficient / Cv
Size Change	Size for Flow	Orifice Bore Size** Venturi and Nozzle Throat Size
	Size for Pressure Loss	
Relief Valve	API RP 520 Part 1	Relief Valve Orifice Size
	ISO 4126 - 1	

**Other process data needed by equipment manufacturers or fabricators are also calculated by FluidFlow. For more information, refer to /Components in the help file accessible by pressing F1 while running FluidFlow.*

***There are two types of orifice components in FluidFlow. "Thin" plate orifices are generally applied for elements used in flow measurement as it applies ISO 5167 equations. "Thick" plate orifice on the other hand is applied when the element is used as flow restriction.*

4. Pipe Sizing

FluidFlow allows you to easily size a pipe or duct through any of these sizing models:

- Economic Velocity
- By Pressure Gradient
- By Velocity

The calculated pipe sizes using any of the sizing models above are reported as either the following:

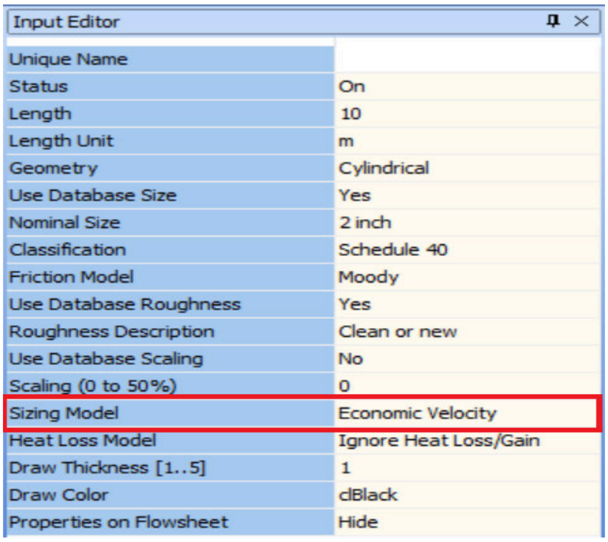
- Exact Economic Size
- Exact Pressure Gradient Size
- Exact Velocity Size

Unlike auto-sizing of other components, the pipe size determined from sizing is not used any further by the solver. Rather, it serves as a suggestion for users to update their defined pipe size.

Thus, changing the pipe sizing model does not impact the calculation results except for the size suggestion as described above.

4.1 Economic Velocity

By default, FluidFlow applies economic velocity as the pipe sizing model. It is a useful guide for pipe sizing, helping engineers develop efficient and economic plant designs.



Input Editor	
Unique Name	
Status	On
Length	10
Length Unit	m
Geometry	Cylindrical
Use Database Size	Yes
Nominal Size	2 inch
Classification	Schedule 40
Friction Model	Moody
Use Database Roughness	Yes
Roughness Description	Clean or new
Use Database Scaling	No
Scaling (0 to 50%)	0
Sizing Model	Economic Velocity
Heat Loss Model	Ignore Heat Loss/Gain
Draw Thickness [1..5]	1
Draw Color	clBlack
Properties on Flowsheet	Hide

Figure 4.1: Pipe Component Input Editor

The economic velocity is calculated using the Generaux equation. This equation takes into account a wide variety of variables such as pipe material, energy and capital costs, fluid properties, and even depreciation and maintenance expenses.

It is represented by this equation:

$$V_{Economic} = \frac{4}{\pi D^2}\left\{\frac{D^{4.84+n} n X E(1+F)[Z+(a+b)(1-\phi)]}{(1+0.794L_e' D)(0.000189 Y K \rho^{0.84}\mu^{0.16})\left[(1+M)(1-\phi)+\frac{ZM}{a'+b'}\right]}\right\}^{\frac{1}{2.84}}$$

Where:

a	Fractional annual depreciation on pipeline, dimensionless
b	Fractional annual maintenance on pipeline, dimensionless
a’	Fractional annual depreciation on pumping installation, dimensionless
b’	Fractional annual maintenance on installation, dimensionless
C	Installed cost of pipeline including fittings, $ / feet
D	Inside pipe diameter, feet
E	Combined fraction efficiency of pump and motor, dimensionless
F	Factor for installation and fitting, dimensionless
K	Energy cost delivered to the motor, $ / kWhr
Le’	Factor for friction in fitting, equivalent length in pipe diameter per length of pipe, 1 / feet
M	Factor to express cost of piping installation in terms of yearly cost of power delivered to the fluid, dimensionless
n	Exponent in pipe-cost equation, dimensionless
P	Installation cost of pump and motor, $ /hp
Q	Fluid Flow, ft3/s
S	Cross sectional area, ft2
V	Velocity, feet / sec
X	Cost of 1 ft of 1 ft diameter pipe, $
Y	Operating days
Z	Fractional rate of return of incremental investment, dimensionless
Φ	Factor for taxes and other expenses, dimensionless
ρ	Flow density, pounds/ ft3
μ	Fluid Viscosity, cP

Note that the information needed to solve the economic velocity is already stored in the database under "Pipe Sizing" and thus, the user is not required to input any additional information.

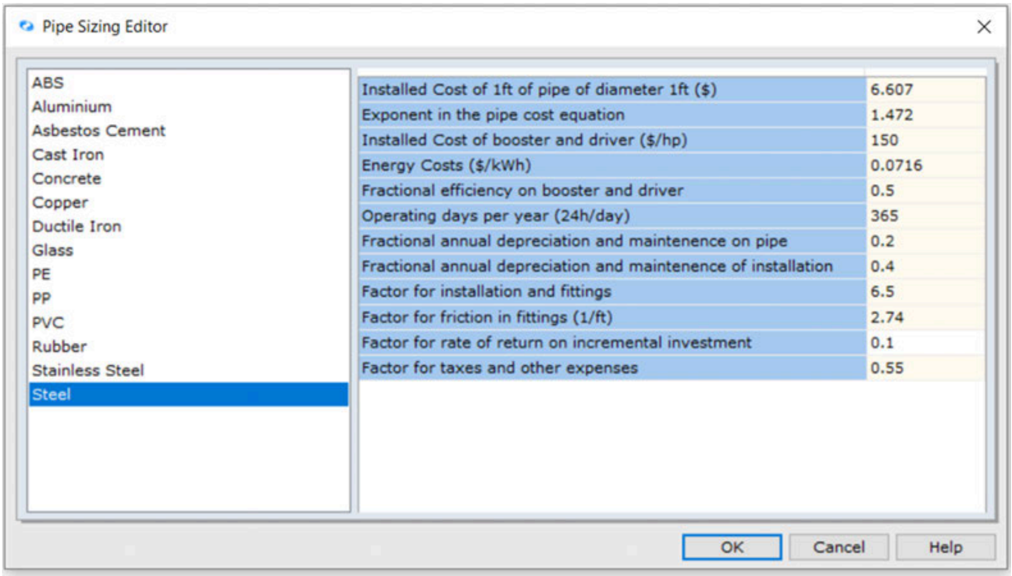


Figure 4.2: Pipe Sizing Database Editor

Nonetheless, the user may update information in the database as required.

When applying the economic velocity model, it is important to consider the type of system being designed. For instance, this criterion should not be used to size pipes where two-phase (liquid-gas) flow is present, where plant operation is intermittent, where materials can degrade at high velocities, or for non-Newtonian flows.

4.2 By Velocity

There are recommended velocities or pressure gradients for many processes with different types of fluids. They are typically based on an organization's experience or a recommendation from a third party's Recognized and Generally Accepted Good Engineering Practice (RAGAGEP). Below is an example of these recommendations given by Sinnot (Chemical Engineering Design).

	Velocity (m/s)	**Pressure differential (kPa/m)**
Liquids, pumped (not viscous)	1-3	0.5

Liquids, gravity flow	-	0.05
Gases and vapours	15-30	0.02 percent of line pressure
High-pressure steam (>8 bar)	30-60	-

Figure 4.3: Recommended velocities and pressure gradients. Sinnott, R. K. (2005). Chemical Engineering Design. Fourth edition. Elsevier Butterworth-Heinemann. Printed in Oxford, UK.

The By velocity sizing model can be used in these cases so that FluidFlow calculates the pipe size that would give the specified velocity with the given flow. In this scenario, FluidFlow also needs as input the desired velocity. This way, FluidFlow can perform the following calculation based on the formula Q = AV:

$$D = \sqrt{\frac{4Q}{\pi V}}$$

Where:
D: inside pipe diameter
V: velocity
Q: volumetric flow rate

4.3 By Pressure Gradient

If desired, FluidFlow can also determine the appropriate pipe size based on a specified pressure gradient and using the Darcy-Weisbach equation:

$$\Delta P/L = \frac{f_D \rho V^2}{2D}$$

Where:
D: inside pipe diameter
V: velocity
ρ: density
f_D: Darcy friction factor
ΔP/L: Pressure gradient

4.4 Limitations of pipe sizing

Applying the suggested size from pipe sizing models is not mandatory, it only serves as a guide for users on developing their designs that would meet their established criteria.

It is also worth noting that pipe sizing criteria can be overruled under certain circumstances. It is up to the user's engineering judgement to determine what these situations are.

Some instances where a pipe sizing criteria is disregarded are the following:

- The suggested pipe size would promote a hazardous or unsafe condition to personnel, assets or nearby communities.
- Applying the suggested size would cause two phase flow issues on fluid transport equipment.
- The resulting pipe size could delay project completion.

5. Sample Equipment Sizing Models

In this section, examples will be shown where a model is built to demonstrate how to automatically size equipment in FluidFlow.

5.1 Auto Size Pipes

System Description:

We wish to transport 100 m^3/h of SAE 30 Lube oil at 15 °C from Reservoir A to B located 15 meters away from each other.

The delivery of oil will be achieved using carbon steel pipe and through enough pressure gradient from Reservoir A which will be received at Reservoir B at a pressure of 2 barg.

Calculation Objective:

Determine the appropriate carbon steel pipe size for delivering 100 m3/hr of SAE 30 Lube Oil from Reservoir A to B using the following criteria:

- Economic Velocity
- Velocity Criteria (1.2 m/s)
- Pressure Gradient (500 Pa/m)

Modeling Steps:

Step 1: Build a Model of the System

A model of this system has been developed as outlined in Figure 5.1.1. This model has been created using steel pipework.

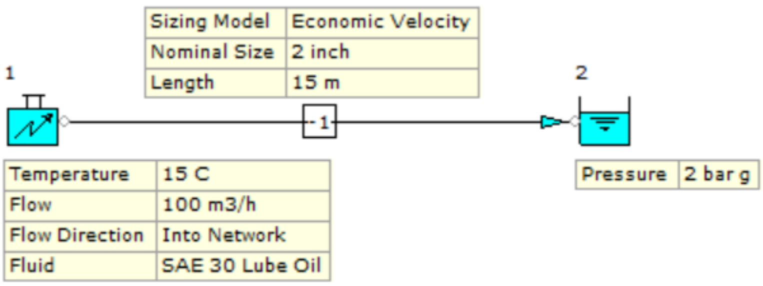
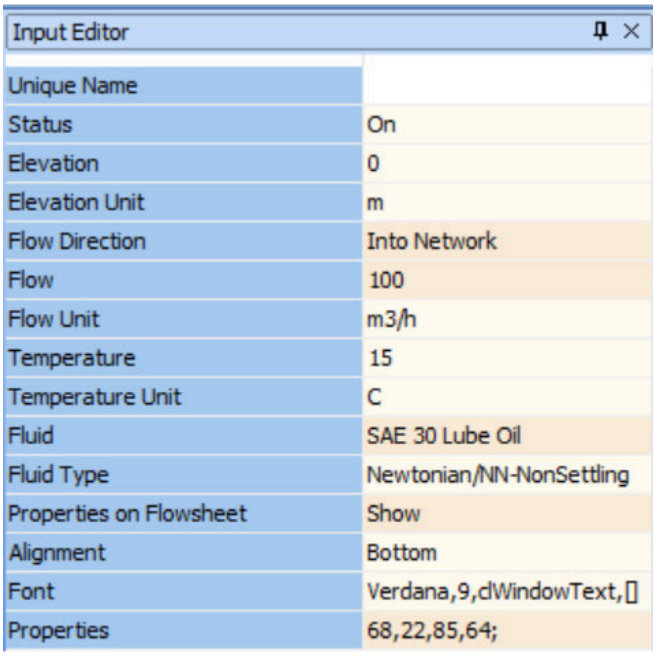


Figure 5.1.1: Water System Model with a 15 m pipe

Step 2: Define the Design Parameters for the nodes

When building a model, the software provides default data for each node/element. This data can be viewed on the Input Editor.

The data entry for the inlet boundary node is shown in Figure 5.1.2.



Input Editor

Unique Name	
Status	On
Elevation	0
Elevation Unit	m
Flow Direction	Into Network
Flow	100
Flow Unit	m3/h
Temperature	15
Temperature Unit	C
Fluid	SAE 30 Lube Oil
Fluid Type	Newtonian/NN-NonSettling
Properties on Flowsheet	Show
Alignment	Bottom
Font	Verdana,9,clWindowText,[]
Properties	68,22,85,64;

Figure 5.1.2: Data Entry for Inlet Boundary

The data entry for the outlet boundary node is shown in Figure 5.1.3.

Input Editor	
Unique Name	
Status	On
Elevation	0
Elevation Unit	m
Pressure Model	Stagnation Pressure
Pressure	2
Pressure Unit	bar g
Temperature	15
Temperature Unit	C
Fluid	water
Fluid Type	Newtonian/NN-NonSettling
Properties on Flowsheet	Show
Alignment	Bottom
Font	Verdana,9,clWindowText,[]
Properties	66;

Figure 5.1.3: Data Entry for Outlet Boundary

Finally, the data entry for the pipe is shown in Figure 5.1.4.

Input Editor	
Unique Name	|
Status	On
Length	15
Length Unit	m
Geometry	Cylindrical
Use Database Size	Yes
Nominal Size	2 inch
Classification	Schedule 40
Friction Model	Moody
Use Database Roughness	Yes
Roughness Description	Clean or new
Use Database Scaling	No
Scaling (0 to 50%)	0
Sizing Model	Economic Velocity
Heat Loss Model	Ignore Heat Loss/Gain
Draw Thickness [1..5]	1
Draw Color	clBlack
Properties on Flowsheet	Show
Alignment	Top
Font	Verdana,9,clWindowText,[]
Properties	79,107,17;

Figure 5.1.4: Data Entry for the Pipe

You may notice that the default Nominal Size for steel pipes is 2 inch and the Classification is Schedule 40. We have simply retained this default data entry as we are going to allow the software to calculate the most economic pipe size for us.

Step 3: Calculate the Model

When we calculate the model, the flow direction is shown for each pipe in the system and we can view the results by clicking the pipe on the flowsheet followed by the Results tab on the Data Palette.

a. Economic Velocity

If you calculate your model at this stage, the economic velocity sizing model has been applied to determine the adequate pipe diameter. Moreover, you will notice the pipe element is highlighted in red which means we have a warning message that we can view on the Messages tab on the Data Palette.

Figure 5.1.5 shows the calculated results with the pipe shown highlighted in red.

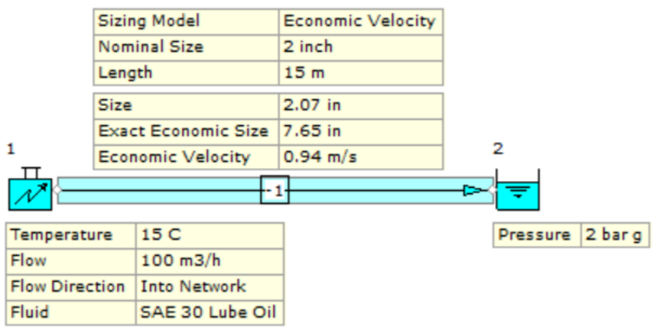


Figure 5.1.5: Calculated System

Figure 5.1.6 shows we have one warning message for the pipe indicating we have a high flow velocity.

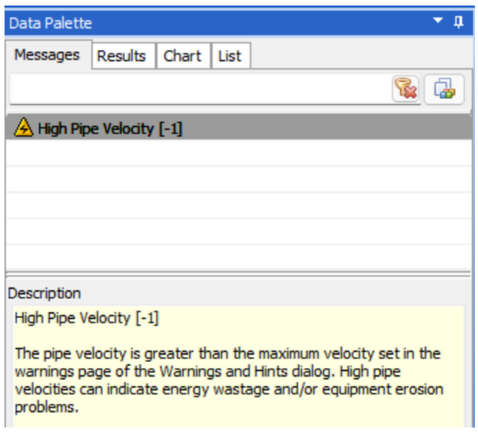
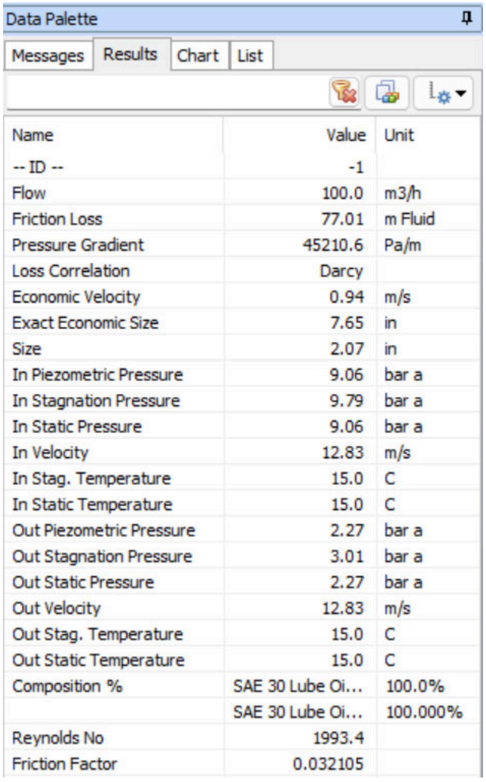


Figure 5.1.6: Warning Messages

We can now review the calculated results in detail by selecting the pipe on the flowsheet followed by the Results tab on the Data Palette.

The results are shown in Figure 5.1.7. We can see that the system has solved with the default pipe size of 2 inch (52.50 mm) which returns a velocity of 12.83 m/s which is considered high for a liquid system. However, we can also see that the calculated Economic Velocity is 0.94 m/s yielding an Exact Economic Size of 7.65 in.

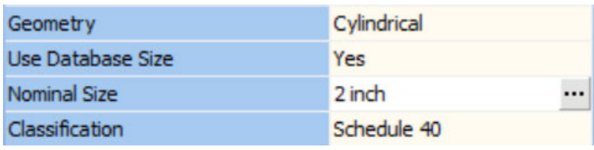


Name	Value	Unit
-- ID --	-1	
Flow	100.0	m3/h
Friction Loss	77.01	m Fluid
Pressure Gradient	45210.6	Pa/m
Loss Correlation	Darcy	
Economic Velocity	0.94	m/s
Exact Economic Size	7.65	in
Size	2.07	in
In Piezometric Pressure	9.06	bar a
In Stagnation Pressure	9.79	bar a
In Static Pressure	9.06	bar a
In Velocity	12.83	m/s
In Stag. Temperature	15.0	C
In Static Temperature	15.0	C
Out Piezometric Pressure	2.27	bar a
Out Stagnation Pressure	3.01	bar a
Out Static Pressure	2.27	bar a
Out Velocity	12.83	m/s
Out Stag. Temperature	15.0	C
Out Static Temperature	15.0	C
Composition %	SAE 30 Lube Oi...	100.0%
	SAE 30 Lube Oi...	100.000%
Reynolds No	1993.4	
Friction Factor	0.032105	

Figure 5.1.7: Pipe Results

As our pipe is just 2.07 in, we can instantly see that we need to increase to a standard pipe size close to 7.65 in to achieve a more economical design velocity near 0.94 m/s.

We can try modeling an 8 inch pipe and review the calculated velocity. From the Input Editor, change the Nominal Size by clicking the grey box on the right-hand side of the field. Figure 5.1.8 gives an overview of the Nominal Size data entry field.



Geometry	Cylindrical
Use Database Size	Yes
Nominal Size	2 inch
Classification	Schedule 40

Figure 5.1.8: Nominal Pipe Size Data Entry Field

This opens the steel pipe database. Expand the 8 Inch field to view the classifications and select schedule 40. Figure 5.1.9 shows the steel pipe database including available sizes.

Figure 5.1.9: Steel Pipe Database

Select OK to confirm this new pipe size. Recalculate the model and you should notice the red warning message has disappeared and the velocity is now 0.86 m/s which is more in line with an economic design.

Note that if capital installation cost is an issue, you can also consider other pipe sizes. For instance, a 6 inch schedule 40 pipe returns a velocity of 1.49 m/s, and a 5 inch pipe a velocity of 2.15 m/s. This allows you to consider efficient design alternatives quickly.

b. By Velocity

If the sizing model is set to "By Velocity", Figure 5.1.10 shows the additional data entries that are required for the pipe in the model.

Sizing Model	By Velocity
Design Velocity	1.2
Velocity Unit	m/s

Figure 5.1.10: By Velocity Pipe Sizing Model Data Entry

Choosing By Velocity allows you to set the design velocity for your pipes (1.2 m/s in this example). When you calculate your piping system, you will see in your results the Exact Velocity Size which is the exact pipe diameter required based on your defined velocity. Figure 5.1.11 shows the results provided when using this pipe sizing approach.

Sizing Velocity	1.20	m/s
Exact Velocity Size	6.76	in
Size	5.05	in

Figure 5.1.11: By Velocity Pipe Sizing Results

We can see at a glance that a pipe size of 6.76 in would be required to achieve this design velocity. We can therefore consider a 6 inch pipe for this system. A 6 inch schedule 40 pipe yields a pressure gradient of 1.49 m/s. Also, you can quickly consider alternative sizes and review the velocity in each case.

c. By Pressure Gradient

If the sizing model is set to "By Pressure Gradient", Figure 5.1.12 shows the additional data entries that are required for the pipe in the model. Note, you can set your desired pressure gradient and associated units.

Sizing Model	By Pressure Gradient
Design Pressure Gradient	500
Pressure Gradient Unit	Pa/m

Figure 5.1.12: By Pressure Gradient Pipe Sizing Model Data Entry

Choosing By Pressure Gradient allows you to set the design pressure gradient for your pipes. When you calculate your piping system, you will see in your results the Exact Pressure Gradient Size which is the exact pipe diameter required based on your defined gradient. Figure 5.1.13 shows the results provided when using this pipe sizing approach (based on a design pressure gradient of 500 Pa/m).

Sizing Pressure Gradient	500.0	Pa/m
Exact Pressure Gradient Size	6.37	in
Size	5.05	in

Figure 5.1.13: By Pressure Gradient Pipe Sizing Results

We can see at a glance that a pipe size of 6.37 inches would be required to achieve this design gradient. We can therefore consider a 6 inch pipe for this system. A 6 inch schedule 40 pipe yields a velocity of 1.49 m/s. Also, you can quickly consider alternative sizes and review the velocity in each case.

Note that when sizing ductwork using any of these three models, the software will calculate the required duct diameter. If you wish to develop a ductwork system with a rectangular geometry, you will need to determine the height and width of the duct required and enter these values into the Input Editor for each section of ductwork. Various literature sources provide tables of equivalent dimensions for circular ductwork, such as the ASHRAE Guides & CIBSE Guide C.

5.2 Control Valve Sizing

Engineers can automatically size flow and pressure control valves to ISA Standard: ISA-75.01.01-2007.

System Description:

The following example case is derived from Example 1, Appendix E of ISA-75.01.01-2007. This system is based on incompressible flow of fluid (water) with a temperature of 363 K, density of 965.4 kg/m^3, inlet and outlet absolute static pressure of 680 and 220 kPa respectively and a flow rate of 360 m^3/h.

Calculation Objective:

Determine the appropriate valve size for the desired conditions of the system.

Modeling Steps:

Step 1: Build a Model of the System.

A model of this system has been developed as outlined in Figure 5.2.1. This model includes a Flow Control Valve (Node 3), as we know the design flow rate for the system based on available data.

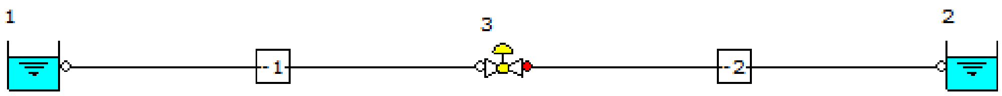
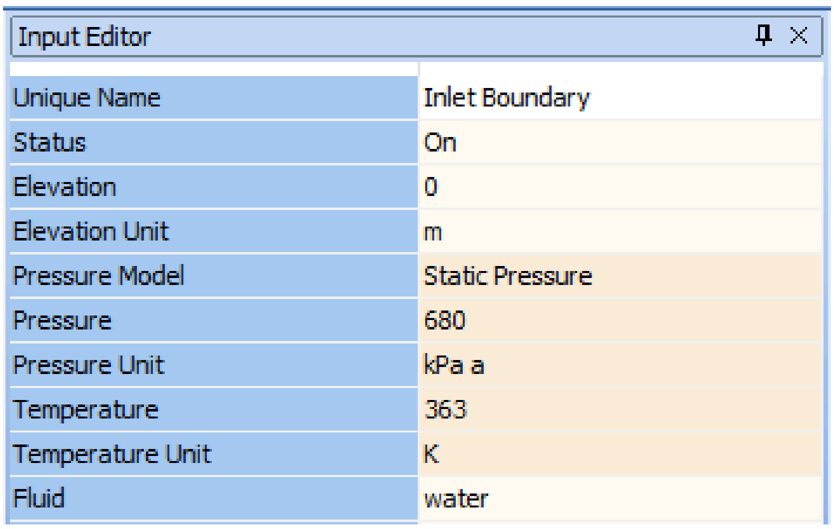


Figure 5.2.1: Flow Control Valve System Model.

Step 2: Define the Design Parameters for the nodes.

The data entry for the inlet boundary node (Node 1) is shown in Figure 5.2.2.



Input Editor

Unique Name	Inlet Boundary
Status	On
Elevation	0
Elevation Unit	m
Pressure Model	Static Pressure
Pressure	680
Pressure Unit	kPa a
Temperature	363
Temperature Unit	K
Fluid	water

Figure 5.2.2: Inlet Boundary – Input Data

The data entry for the outlet boundary node (Node 2) is shown in Figure 5.2.3.

Input Editor	
Unique Name	Outlet Boundary
Status	On
Elevation	0
Elevation Unit	m
Pressure Model	Static Pressure
Pressure	220
Pressure Unit	kPa a
Temperature	363
Temperature Unit	K
Fluid	water

Figure 5.2.3: Outlet Boundary – Input Data

As we know the design flow rate for the system, we can set this parameter at the Flow Control Valve. The data entry for this valve is as shown in Figure 5.2.4.

Input Editor	
Unique Name	Flow Control Valve
Status	On
Elevation	0
Elevation Unit	m
Automatically Size	On
Control Valve Type	Globe (Xt = 0.75)
Design Flow	360
Flow Unit	m3/h
Discharge Pipe (RED)	-2
Heat Loss Model	Ignore Heat Loss/Gain

Figure 5.2.4: Control Valve – Input Data

As we can see in Figure 5.2.4, the value for Xt is set to 0.75 as per the ISA Standard example. The connected pipes are also set to 150 mm in line with the worked example.

Step 3: Calculate the Model

When we solve this system, FluidFlow calculates a Kv of 165.086 m^3/h/bar whereas the hand calculation notes a Cv value of 165 m^3/h/bar. The results are almost identical.

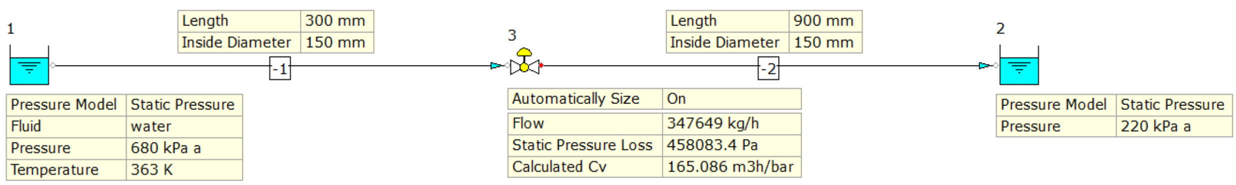
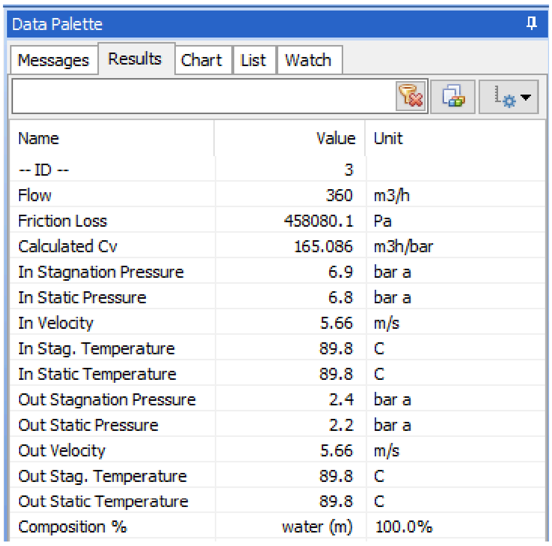


Figure 5.2.5: Automatically Sized Solved Model

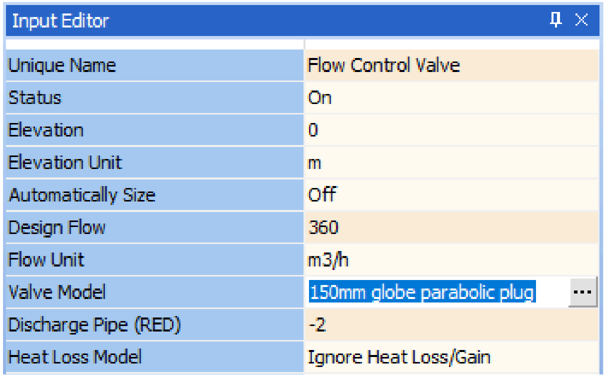
An overview of the FluidFlow results is shown in Figure 5.2.6.



Name	Value	Unit
-- ID --	3	
Flow	360	m3/h
Friction Loss	458080.1	Pa
Calculated Cv	165.086	m3h/bar
In Stagnation Pressure	6.9	bar a
In Static Pressure	6.8	bar a
In Velocity	5.66	m/s
In Stag. Temperature	89.8	C
In Static Temperature	89.8	C
Out Stagnation Pressure	2.4	bar a
Out Static Pressure	2.2	bar a
Out Velocity	5.66	m/s
Out Stag. Temperature	89.8	C
Out Static Temperature	89.8	C
Composition %	water (m)	100.0%

Figure 5.2.6: Control Valve Results.

You can also model vendor specific valves in your model. Setting the Automatically Size function to OFF, allows you to model a manufacturer's valve from the database, as shown in Figure 5.2.7.



Unique Name	Flow Control Valve
Status	On
Elevation	0
Elevation Unit	m
Automatically Size	Off
Design Flow	360
Flow Unit	m3/h
Valve Model	150mm globe parabolic plug
Discharge Pipe (RED)	-2
Heat Loss Model	Ignore Heat Loss/Gain

Figure 5.2.7: Vendor Control Valve Selection

Figures 5.2.8 and 5.2.9 show the results of a calculation that utilizes the globe parabolic plug valve as described in the ISA Standard. The results clearly show the % opening of the valve based on the design flow rate, in this case, the control valve would be 58.96% open in order to achieve the design conditions.

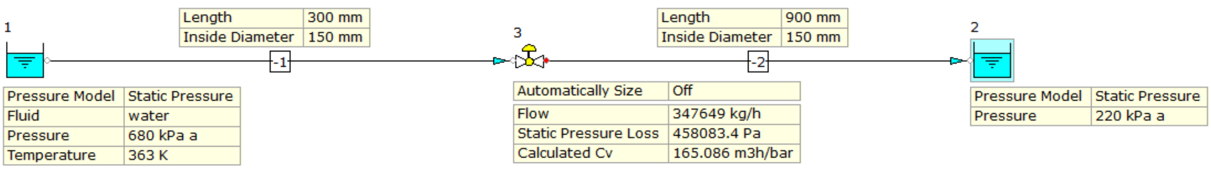


Figure 5.2.8: Globe Parabolic Plug Valve Solved Model

Name	Value	Unit
-- ID --	3	
Flow	360	m3/h
Friction Loss	458080.1	Pa
Calculated Cv	165.086	m3h/bar
Valve Opening (%)	58.96	
In Stagnation Pr...	6.9	bar a
In Static Pressure	6.8	bar a
In Velocity	5.66	m/s
In Stag. Temper...	89.8	C
In Static Temper...	89.8	C
Out Stagnation ...	2.4	bar a
Out Static Pressure	2.2	bar a
Out Velocity	5.66	m/s
Out Stag. Temp...	89.8	C
Out Static Temp...	89.8	C
Composition %	water (m)	100.0%

Figure 5.2.9: Globe Parabolic Plug Valve Results

5.3 Pump Sizing

Pumps can be automatically sized, based on either a design flow rate or design pressure rise across the system. The example in Figure 5.3.1 demonstrates both approaches.

System Description:

It is desired to transport water at 15 ºC and 1 atm pressure to an outlet with a design absolute pressure of 1 bar, and an elevation of 9.5 m, via a network of 50 m total length, 50 mm pipes.

Calculation Objective:

Obtain the appropriate pump specifications to transport the fluid, and compare the different sizing methods.

Modeling Steps:

Step 1: Build a Model of the System

Two models have been developed as shown in Figure 5.3.1 to compare the sizing methods. The pump of the top model will be designed for flow, and the pump on the bottom model will be designed for pressure rise.

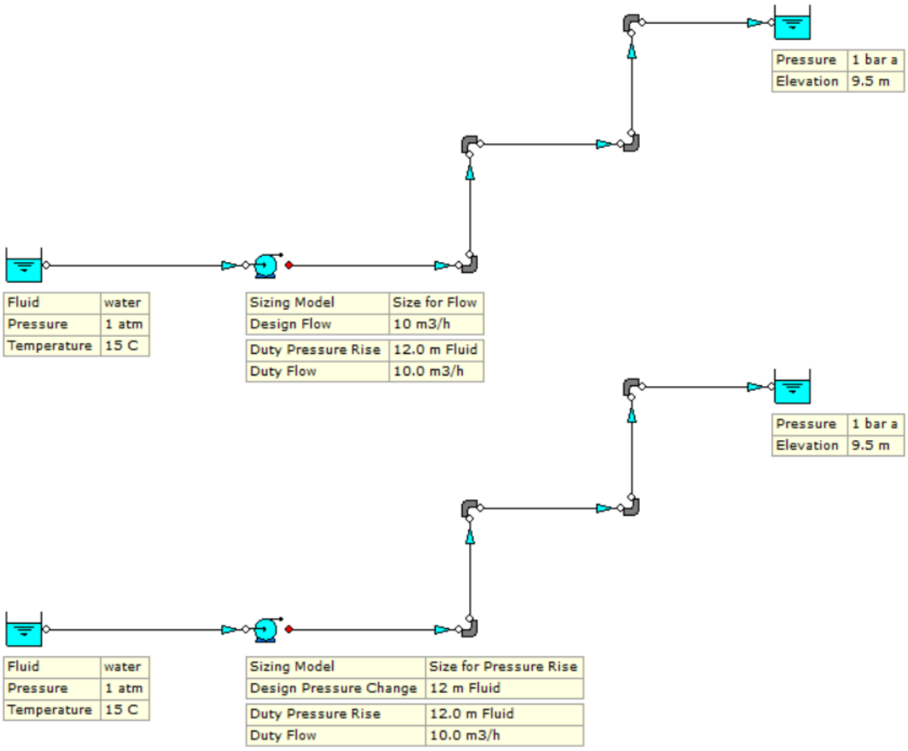
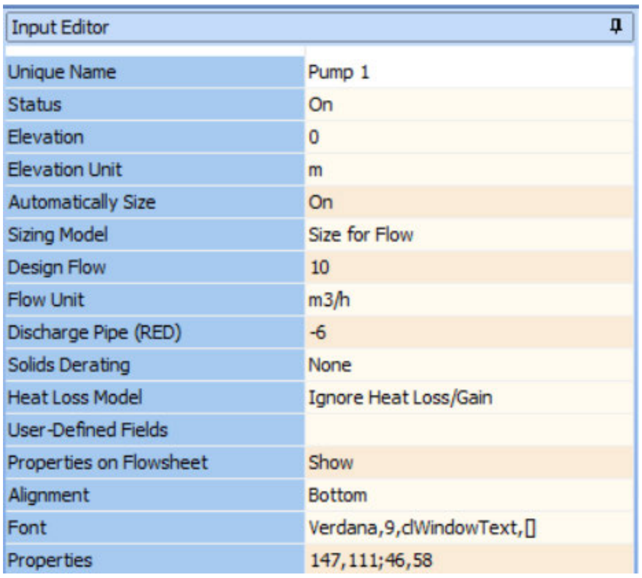


Figure 5.3.1: Pump Sizing Models

Step 2: Define the Design Parameters

The pump in the top model is sized based on a design flow rate of 10 m^3/h, as shown in Figure 5.3.2.



Input Editor	
Unique Name	Pump 1
Status	On
Elevation	0
Elevation Unit	m
Automatically Size	On
Sizing Model	Size for Flow
Design Flow	10
Flow Unit	m3/h
Discharge Pipe (RED)	-6
Solids Derating	None
Heat Loss Model	Ignore Heat Loss/Gain
User-Defined Fields	
Properties on Flowsheet	Show
Alignment	Bottom
Font	Verdana,9,clWindowText,[]
Properties	147,111;46,58

Figure 5.3.2: Pump sized for flow

Based on the design conditions, the calculated duty pressure rise is 12 m fluid.

The bottom model is an identical match, however, the pump sizing criteria is set to Size for Pressure Rise, which is set to 12 m fluid, as shown in Figure 5.3.3.

Input Editor	
Unique Name	Pump 2
Status	On
Elevation	0
Elevation Unit	m
Automatically Size	On
Sizing Model	Size for Pressure Rise
Design Pressure Change	12
Pressure Change Unit	m Fluid
Discharge Pipe (RED)	-7
Solids Derating	None
Heat Loss Model	Ignore Heat Loss/Gain
User-Defined Fields	
Properties on Flowsheet	Show
Alignment	Bottom
Font	Verdana,9,clWindowText,[]
Properties	150,111;46,58

Figure 5.3.3: Pump sized for pressure rise

Step 3: Calculate the Model

When the simulation is completed, we can see the resultant flow rate calculated is 10 m^3/h which is identical to the first system.

An overview of some pump results for automatically sized pumps is shown in Figure 5.3.4.

Data Palette — Messages | Results | Chart | List | Watch

Name	Value	Unit
-- ID --	7	
Duty Flow	10	m3/h
Duty Pressure Rise	12.0	m Fluid
Duty NPSH Available	9.9	m Fluid
In Stagnation Pressure	1.0	bar a
In Static Pressure	1.0	bar a
In Stag. Temperature	15.0	C
In Static Temperature	15.0	C
Out Stagnation Pressure	2.2	bar a
Out Static Pressure	2.2	bar a
Out Velocity	1.41	m/s
Out Stag. Temperature	15.0	C
Out Static Temperature	15.0	C
Out Viscosity	1.137	cP
Composition %	water (m)	100.0%

Figure 5.3.4: Automatically Sized Pump Results.

Note, that when we model a vendor pump in our system, the software also calculates the duty power requirement, efficiency, and $NPSH_R$ as presented in Figure 5.3.5. You can also model changes in pump speed and impeller diameter thus helping you optimize your system performance.

Note, FluidFlow allows you to automatically size centrifugal pumps, fans and compressors based on a design flow rate or duty pressure rise. PD pumps and Rotating PD pumps can be automatically sized based on a design system flow rate.

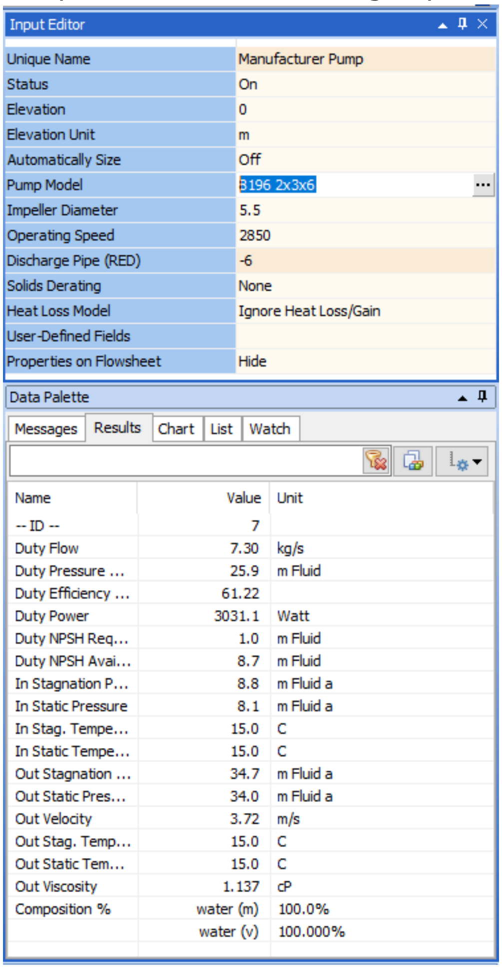


Figure 5.3.5: Manufacturer Pump Example

5.4 Orifice Plate Sizing

With FluidFlow users can also size orifice plates for a desired flow or pressure drop based on ISO 5167. A comparison study has been completed on example 8-10 found in Cengel, Y. and Cimbala, J. (2018). Fluid Mechanics. Fundamentals and Applications. 4th Edition. McGraw-Hill Education Global Holdings LLC.

System Description: The design pipe diameter is 4 cm with an orifice size of 3 cm. The benchmark hand calculation is based on methanol at 20°C, a fluid density of 788.4 kg/m^{3,} and a viscosity of 0.5857 cP. The hand calculation also quotes a flow rate of 3.09 L/s. Based on ISO 5167 we can calculate the orifice plate pressure loss to create the model: 11041.92 Pa.

Calculation Objective:

Determine the optimal size orifice, applying the FluidFlow sizing models, and compare those results with the hand calculations.

Modeling Steps:

Size for Flow

Step 1: Build a Model of the System

Figure 5.4.1 shows the model of the system developed in FluidFlow.

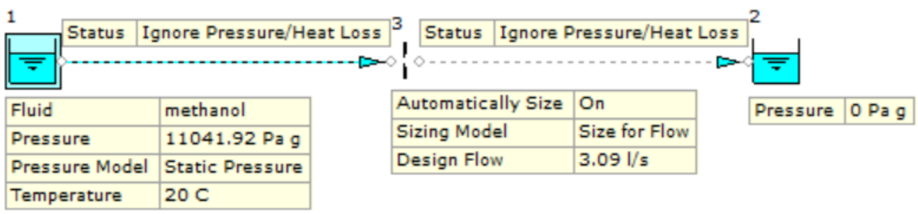
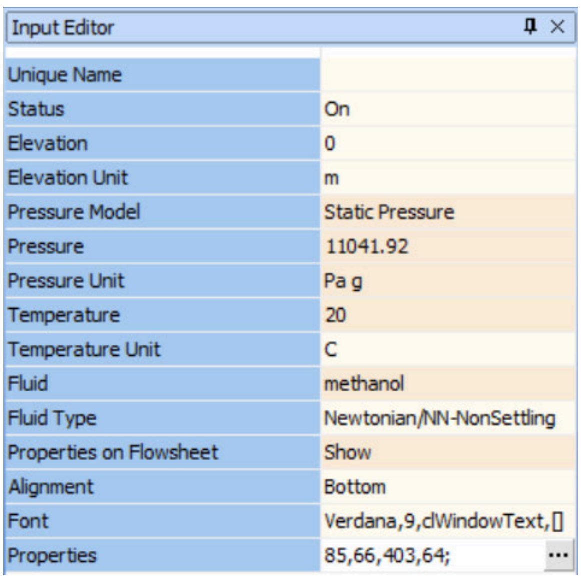


Figure 5.4.1: Size for flow model

Step 2: Define the Design Parameters

The orifice plate has been automatically sized (see Figure 5.4.1) based on the design flow rate of 3.09 L/s.

The data entry for the inlet boundary node (Node 1) is shown in Figure 5.4.2.



Input Editor

Unique Name	
Status	On
Elevation	0
Elevation Unit	m
Pressure Model	Static Pressure
Pressure	11041.92
Pressure Unit	Pa g
Temperature	20
Temperature Unit	C
Fluid	methanol
Fluid Type	Newtonian/NN-NonSettling
Properties on Flowsheet	Show
Alignment	Bottom
Font	Verdana,9,clWindowText,[]
Properties	85,66,403,64;

Figure 5.4.2: Inlet Boundary - Input Data

The data entry for the outlet boundary node (Node 2) is shown in Figure 5.4.3.

Input Editor	
Unique Name	
Status	On
Elevation	0
Elevation Unit	m
Pressure Model	Stagnation Pressure
Pressure	0
Pressure Unit	Pa g
Temperature	15
Temperature Unit	C
Fluid	water
Fluid Type	Newtonian/NN-NonSettling
Properties on Flowsheet	Show
Alignment	Bottom
Font	Verdana,9,clWindowText,[]
Properties	66;

Figure 5.4.3: Outlet Boundary - Input Data

The data entry for the thin sharp edged orifice node (Node 3) is shown in Figure 5.4.4.

Input Editor	
Unique Name	
Status	On
Elevation	0
Elevation Unit	m
Automatically Size	On
Sizing Model	Size for Flow
Design Flow	3.09
Flow Unit	l/s
Orifice Equation	ISO 5167
Heat Loss Model	Ignore Heat Loss/Gain
Properties on Flowsheet	Show
Alignment	Bottom
Font	Verdana,9,clWindowText,[]
Properties	17,147,111;

Figure 5.4.4: Orifice Plate - Input Data

Notice that, as the model is being used to calculate only the orifice plate, the pressure loss through the pipes has been ignored.

Step 3: Calculate the Model

When the system is solved, FluidFlow calculates the size of the orifice as 30 mm (see Figure 5.4.5).

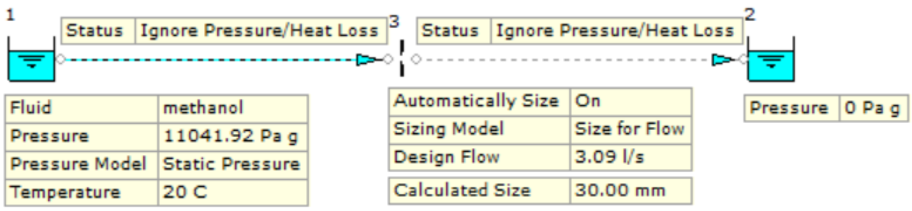


Figure 5.4.5: Calculated Model in FluidFlow

Size for pressure loss

Step 1: Build a Model of the System

Figure 5.4.6 shows the model of the system developed in FluidFlow.

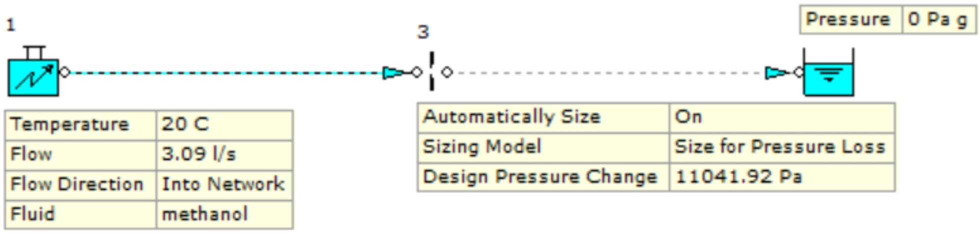
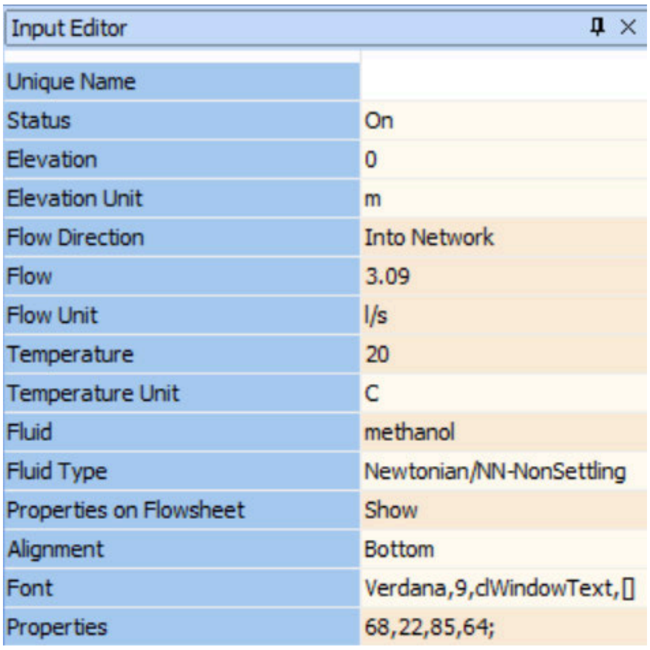


Figure 5.4.6: Size for pressure loss model

Step 2: Define the Design Parameters

The data entry for the inlet boundary node (Node 1) is shown in Figure 5.4.7.



Input Editor

Unique Name	
Status	On
Elevation	0
Elevation Unit	m
Flow Direction	Into Network
Flow	3.09
Flow Unit	l/s
Temperature	20
Temperature Unit	C
Fluid	methanol
Fluid Type	Newtonian/NN-NonSettling
Properties on Flowsheet	Show
Alignment	Bottom
Font	Verdana,9,clWindowText,[]
Properties	68,22,85,64;

Figure 5.4.7: Inlet Boundary - Input Data

The data entry for the outlet boundary node (Node 2) is shown in Figure 5.4.8.

Input Editor	
Unique Name	
Status	On
Elevation	0
Elevation Unit	m
Pressure Model	Stagnation Pressure
Pressure	0
Pressure Unit	Pa g
Temperature	15
Temperature Unit	C
Fluid	water
Fluid Type	Newtonian/NN-NonSettling
Properties on Flowsheet	Show
Alignment	Top
Font	Verdana,9,clWindowText,[]
Properties	66;

Figure 5.4.8: Outlet Boundary - Input Data

The data entry for the thin sharp edged orifice node (Node 3) is shown in Figure 5.4.9.

Input Editor	
Unique Name	
Status	On
Elevation	0
Elevation Unit	m
Automatically Size	On
Sizing Model	Size for Pressure Loss
Design Pressure Change	11041.92
Pressure Change Unit	Pa
Orifice Equation	ISO 5167
Heat Loss Model	Ignore Heat Loss/Gain
Properties on Flowsheet	Show
Alignment	Bottom
Font	Verdana,9,clWindowText,[]
Properties	17,150,111;

Figure 5.4.9: Orifice Plate - Input Data

Step 3: Calculate the Model

When we solve this system, FluidFlow calculates the size for the orifice as 30 mm (see Figure 5.4.10).

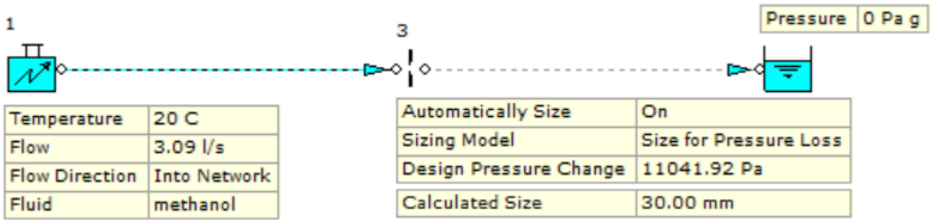


Figure 5.4.10: Calculated Model in FluidFlow

5.5 Balancing Flow Distribution through Autosize

This example involves designing a cooling water distribution system to a bank of heat exchangers where we shall use orifice plates to balance the flow distribution. We shall also use the auto-sizing functions to develop the system design and size the pipes, pump, and orifice plates.

System Description:

It is desired to provide a balanced cooling water flow to four shell and tube heat exchangers HE1, HE2, HE3 and HE4. The size of the heat exchangers has already been determined from the process requirement and is summarized in Table 5.5.1.

Table 5.5.1: Heat Exchanger Data.

Item	Mass flow (kg/s)	Heat Load (kW)	Tube Length (m)	Tube Diameter (mm)	Number of Tubes
HE1	2.969	370	4	20	20
HE2	2.006	250	4	20	20
HE3	2.969	370	4	20	20
HE4	2.006	250	4	20	20

The cooling water is to flow through the heat exchangers and the design system inlet temperature will be 15 °C. The design temperature rise of the cooling water across each heat exchanger is 30 °C. The elevation of all elements in this model is zero. The design duty pressure rise for this system is also known as 2.5 bar.

Calculation Objective:

Determine the appropriate size and operations conditions for each of the components in the cooling system.

Modeling Steps:

Step 1: Build a Model of the System

1. Place two known pressure boundary nodes on the flow sheet.

2. Using the steel pipe element, connect the known pressure boundary nodes together as per Figure 5.5.1.
3. Add the branch pipe connections, ensuring the branch at each tee junction is assigned correctly.
4. Insert the heat exchangers into each branch line.
5. Insert thin orifice plates into the branch pipe connection serving each heat exchanger
6. Insert the Centrifugal Pump at the pipe connected to the known pressure boundary node 1.

When complete, the model should appear as per Figure 5.5.1.

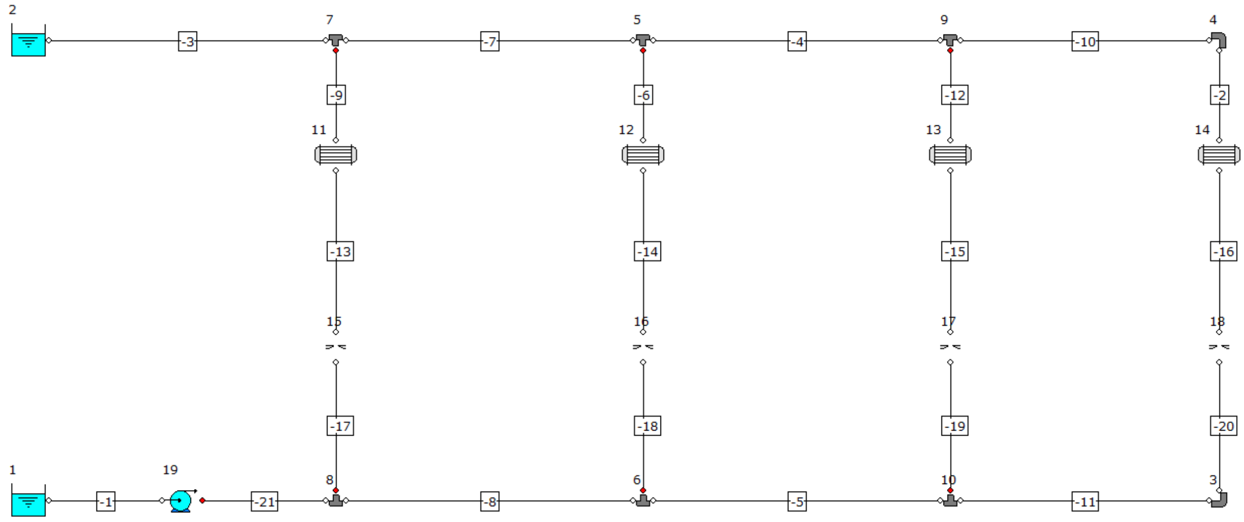


Figure 5.5.1: Base Model.

Step 2: Define the Design Parameters

Each element placed on the flowsheet is provided with default data which the engineer can easily edit based on known design parameters. In this case, we will define the pipe lengths in Table 5.5.2. We will accept the default data provided by FluidFlow for the boundary nodes.

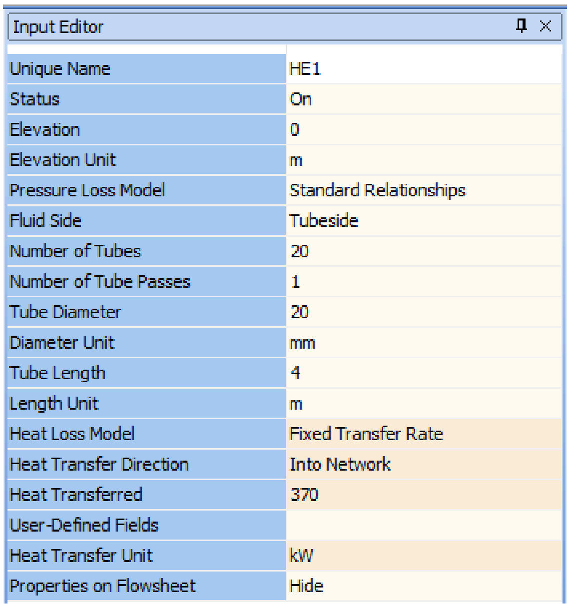
Table 5.5.2: Pipe Length Data

Pipe User Number	Pipe Length (m)	Pipe User Number	Pipe Length (m)
-17	5	-2	5
-13	5	-11	5
-9	5	-10	5
-18	5	-5	2.5
-14	5	-4	2.5
-6	5	-8	1.25
-19	5	-7	1.25
-15	5	-3	1.25
-12	5	-21	1.25

Pipe User Number	Pipe Length (m)	Pipe User Number	Pipe Length (m)
-20	5	-1	1
-16	5		

Select all the heat exchangers at once by holding the SHIFT key and left mouse-clicking on each. All heat exchangers should now be highlighted on the flowsheet. From the Input Editor, set the Heat Loss Model to Fixed Transfer Rate, the heat transfer direction to Into the Network and the Heat Transfer Unit to kW. We have now set all common parameters for the heat exchangers in one step.

The next step is to define the heat load for each heat exchanger (see Table 5.5.1). Figure 5.5.2 shows the input data for the heat exchanger HE1 (Node 11).

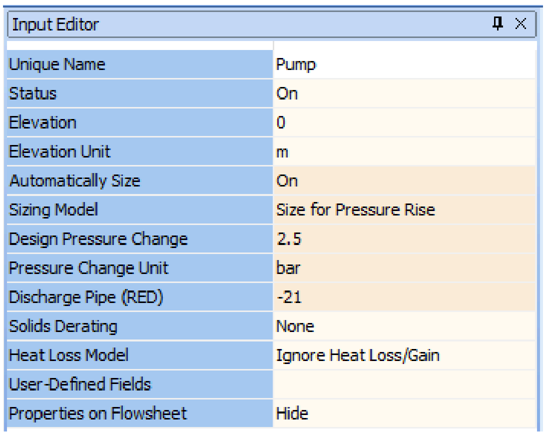


Input Editor

Unique Name	HE1
Status	On
Elevation	0
Elevation Unit	m
Pressure Loss Model	Standard Relationships
Fluid Side	Tubeside
Number of Tubes	20
Number of Tube Passes	1
Tube Diameter	20
Diameter Unit	mm
Tube Length	4
Length Unit	m
Heat Loss Model	Fixed Transfer Rate
Heat Transfer Direction	Into Network
Heat Transferred	370
User-Defined Fields	
Heat Transfer Unit	kW
Properties on Flowsheet	Hide

Figure 5.5.2: HE1 input editor

The design pump pressure rise is 2.5 bar. We can therefore set the centrifugal pump to Automatically Size from the Input Editor. In doing so, we have two sizing options available: *Size for Flow* and *Size For Pressure Rise*. Select *Size For Pressure Rise* and define the *Design Pressure Change* as 2.5 bar, as shown in Figure 5.5.3.

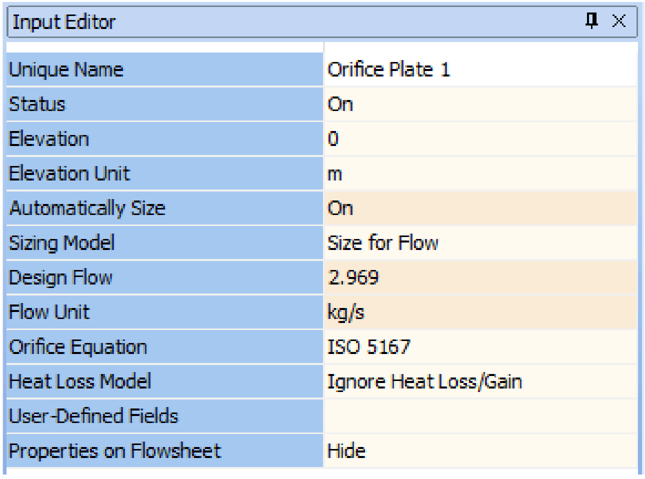


Input Editor

Unique Name	Pump
Status	On
Elevation	0
Elevation Unit	m
Automatically Size	On
Sizing Model	Size for Pressure Rise
Design Pressure Change	2.5
Pressure Change Unit	bar
Discharge Pipe (RED)	-21
Solids Derating	None
Heat Loss Model	Ignore Heat Loss/Gain
User-Defined Fields	
Properties on Flowsheet	Hide

Figure 5.5.3: Pump for pressure rise

Since we can auto-size our components, we can define the design flow rate at each orifice plate by setting the *Automatically Size* option on the Input Editor to On. We now have two sizing models to choose from, Size for Flow and Size for Pressure Loss. As we know the flow rate, select Size for Flow. The next step is to enter the mass flow rates noted in Table 5.5.1 for each orifice plate.


Input Editor

Unique Name	Orifice Plate 1
Status	On
Elevation	0
Elevation Unit	m
Automatically Size	On
Sizing Model	Size for Flow
Design Flow	2.969
Flow Unit	kg/s
Orifice Equation	ISO 5167
Heat Loss Model	Ignore Heat Loss/Gain
User-Defined Fields	
Properties on Flowsheet	Hide

Figure 5.5.4: Orifice plate 1 sizing

When using the auto size function, it is important not to over-constrain the model as this could lead to model convergence difficulties. For instance, in this example, we could have set the sizing model for the pump to Size for Flow. As we had effectively already defined the total flow in the system by defining the flow rate at each orifice plate, we would have duplicated the flow in the system, obtaining the warning message shown below in Figure 5.5.5. Hence, it is best to avoid duplicating design parameters when developing your model, such as what we did by setting the pump to Size for Pressure Rise.

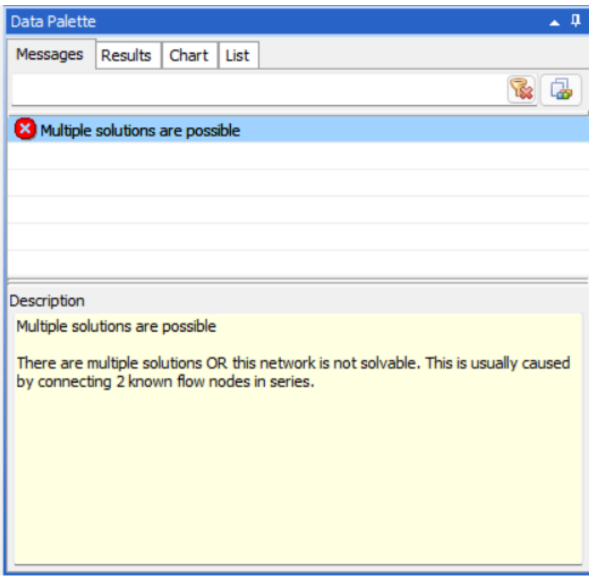


Figure 5.5.5: Multiple solutions warning message

The Sizing Model for pipes is Economic Velocity by default. This means that FluidFlow will determine the Exact Economic Size pipe diameter based on the calculated Economic Velocity.

As we wish to develop an efficient system design, we are going to retain the Economic Velocity sizing model.

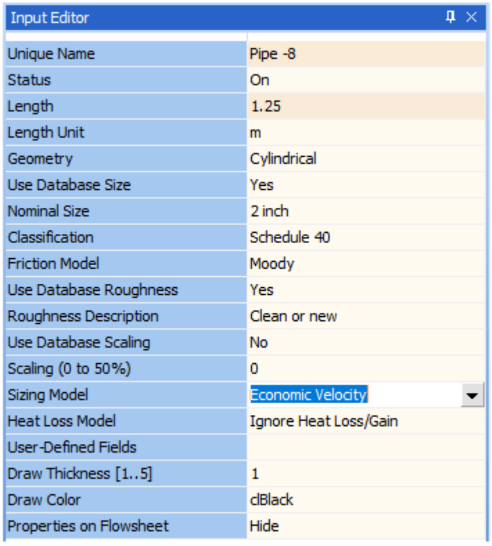


Figure 5.5.6: Economic Velocity Selection

Step 3: Calculate the Model

We are now in a position to calculate the model. The solved system should appear as set out in Figure 5.5.7.

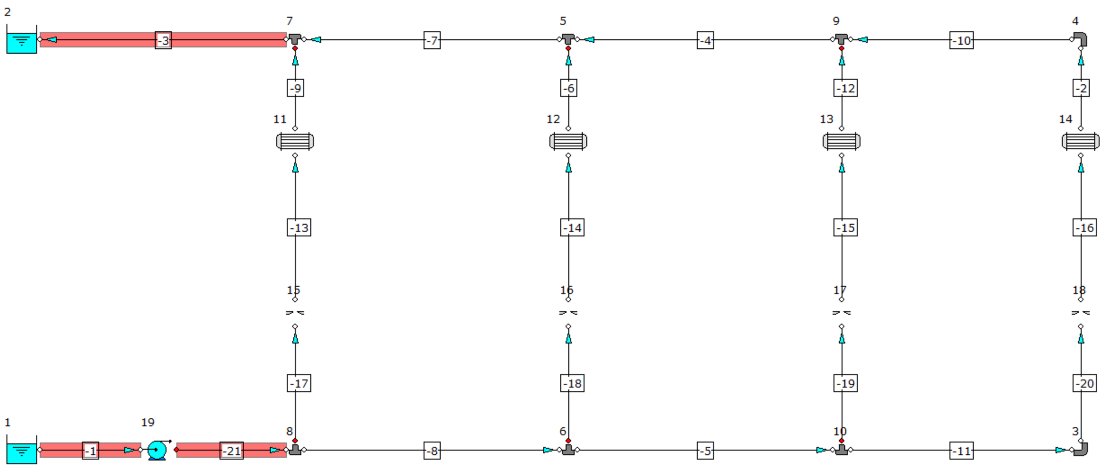


Figure 5.5.7: Solved Model – Default 2 Inch Pipes

The flow distribution has been shown and if we view the results for any of the four heat exchangers, we can see that the inlet temperature is 15°C and the outlet temperature is 45°C (based on our design 30°C temperature rise).

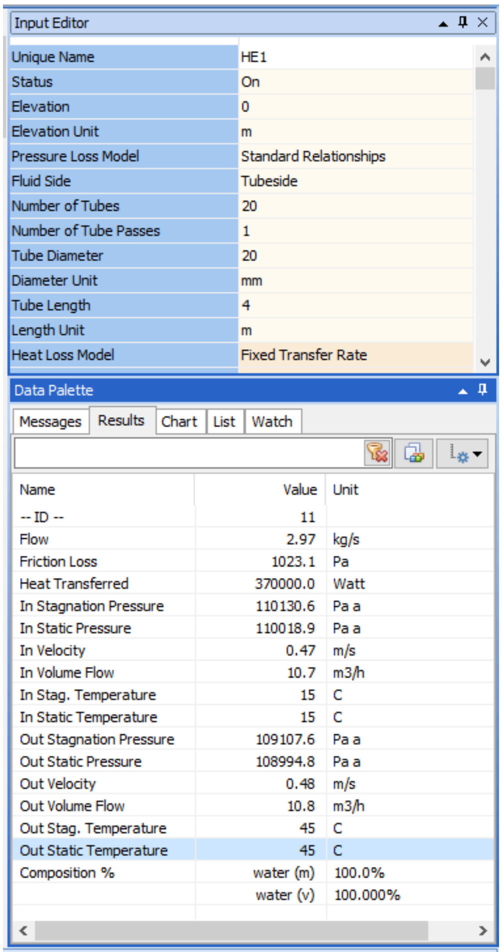


Figure 5.5.8: HE1 Results

Note that we also have three warning messages indicating high velocities in the pipelines highlighted in RED.

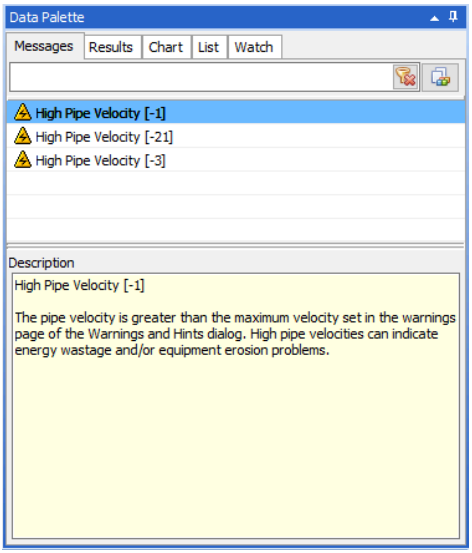


Figure 5.5.9: High Pipe Velocity Warning Messages

A quick check on the results for each of the pipes with a high velocity warning indicates velocities in the range of 4.5 m/s, which is considered high. We therefore need to review the pipe diameter. The diameter of each of these pipes is the default value of 2 inch, which is 52.5 mm. FluidFlow has determined an economic pipe size of approx. 101 mm for each of these three pipes.

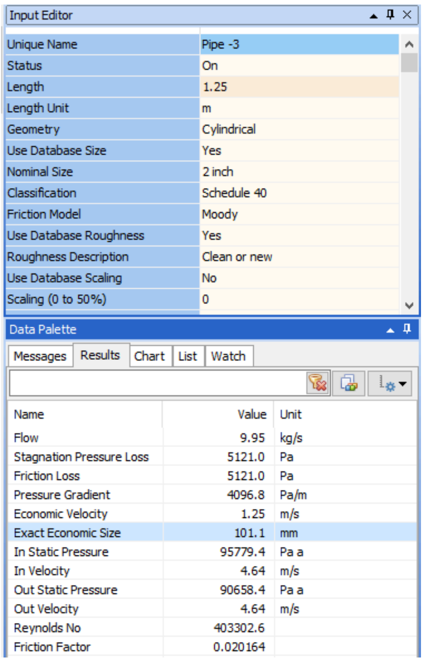


Figure 5.5.10: Economic Size

We therefore need to select the next closest standard size match. Let's try a 4 inch schedule 40 pipe. You can multi-select the three pipes by holding the SHIFT key and left mouse-clicking on each pipe. From the Input Editor, access the pipes database and change the pipe to 4 inch schedule 40 pipe. Press Calculate to refresh the results for the system.

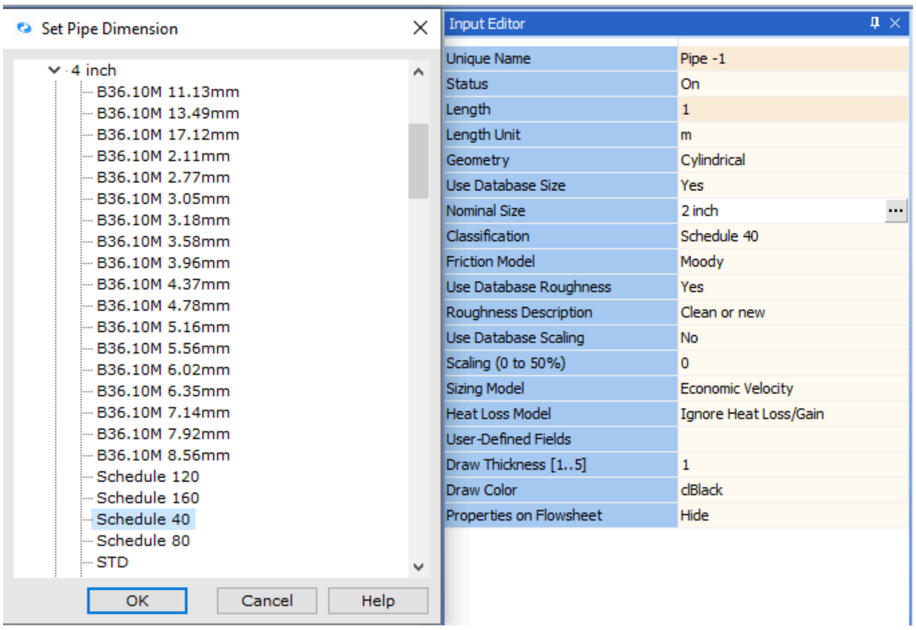


Figure 5.5.11: Resizing pipes

If we view the results for the orifice plates, we can see that FluidFlow has automatically determined the K value and the size of the orifice required to provide the desired mass flow rate. The pump duty has also been established for us as 9.95 kg/s @ 2.5 bar (25.5 m fluid).

The warnings have also been corrected and as such, it can be considered that the system is now a relatively efficient design.

Based on the data entry, FluidFlow has therefore automatically sized the piping, pump and orifice plates for us, thus simplifying the design process and reducing design time considerably.

The final design should appear as set out in Figure 5.5.12.

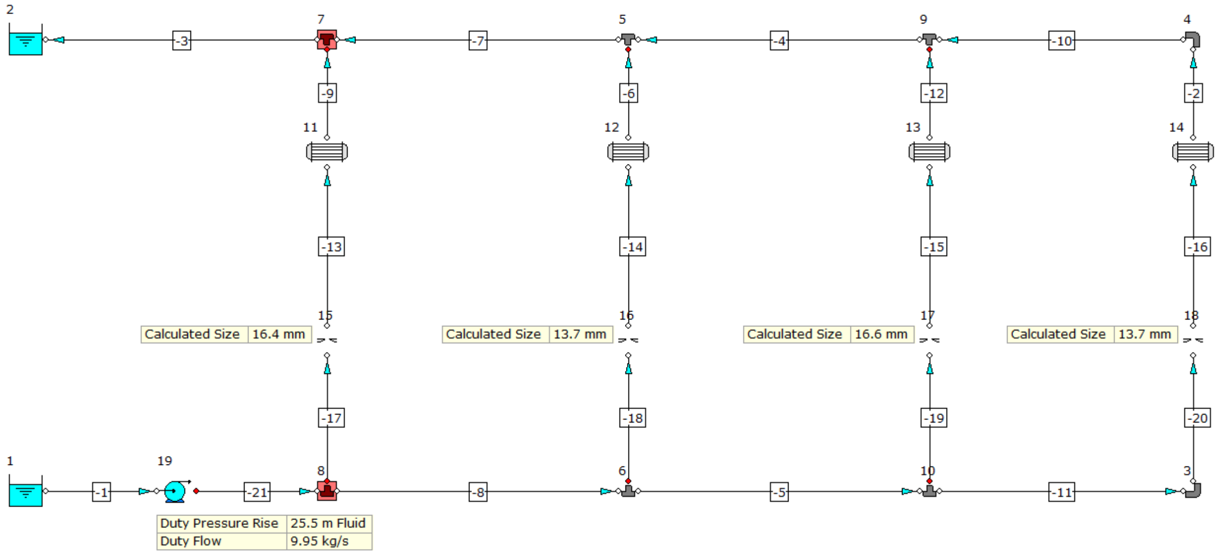


Figure 5.5.12: Solved Model – Updated Pipe Diameters

FluidFlow

Testimonial

London Marine Consultants (LMC) specialises in the design of external turret mooring systems for FPSOs, FSRUs and FSOs. FluidFlow was used during the engineering phase of different projects to ensure that the turret's complex swivel system and its surrounding pipe network were the most adequate and met all of the client's requirements (i.e. pipe size, pressure drop, velocity etc).

FluidFlow was the perfect software for this job:

- We had a powerful tool that allowed us to model a system as complex as a swivel and optimize its surrounding pipe network
- The interface is user-friendly and yet very accurate (large database, various correlations can be used, modelling of multi-phase flow, results can be exported and treated in Excel)
- And finally the FluidFlow team is very knowledgeable and always prompt to respond whenever needed.

Steph Thoni - London Marine

Get in Touch

Start your free 14-day trial today

fluidflowinfo.com

sales@fluidflowinfo.com
support@fluidflowinfo.com

Toll free: +1 888 711 3051
Worldwide: +44 28 7127 9227