Discharging Tank

Introduction

This tutorial model illustrates how to calculate the pressure drop and initial flow rate in a pipe system connected to water tank. The Pipe Flow interface contains ready-to-use friction models accounting for the surface roughness of pipes as well as pressure losses in bends and valves.

Model Definition

Water from a tank flows through a total of 105 m of pipe to be discharged through an open ball valve. The pipes are 15 cm in diameter and made out of galvanized iron. The water level is 10 m above the point of discharge.

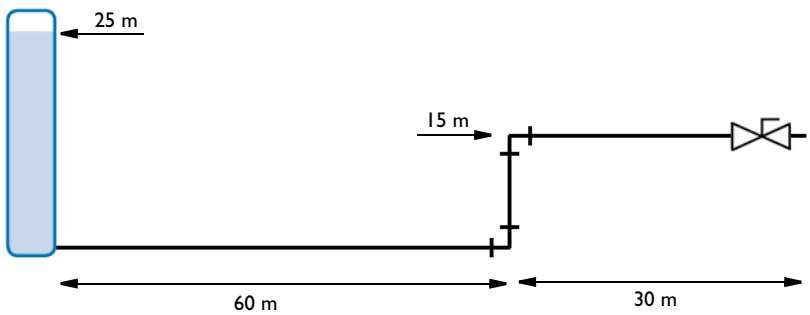

Figure 1: Water flows through a pipe system with two 90° bends and discharges through an open ball valve.

The model example is taken from Ref. 1.

THE MOMENTUM EQUATION

Inside a stretch of pipe section, the momentum balance solved is:

$$\nabla p = -f_{\mathrm{D}}\frac{\rho}{2d_{\mathrm{h}}}\mathbf{u}|\mathbf{u}| + \mathbf{F} \tag{1}$$

and the continuity equation

$$\nabla \cdot (A\rho\mathbf{u}) = 0 \tag{2}$$

The term on the left-hand side of Equation 1 is the pressure gradient along the tangential direction (flow direction) of a pipe stretch. The first term on the right-hand side represents the pressure drop due to viscous shear. f_D is the Darcy friction factor, d_h is the hydraulic diameter and $\mathbf{u}$ is the velocity mean value across a pipe cross section. $\mathbf{F}$ is a volume force term (SI unit: N/m^3), in this case used to account for gravity. ρ and A in Equation 2 are fluid density (kg/m^3) and cross section area (m^3), respectively. To find out more about these equations and variables, please refer to the section Theory for the Pipe Flow Interface in the *Pipe Flow Module Users's Guide*.

Expressions for the Darcy Friction Factor

The Pipe Flow interface provides a library of built-in expressions for the Darcy friction factor, f_D.

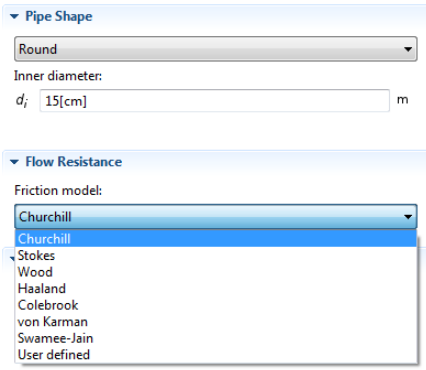

Figure 2: Select from different predefined Friction models in the Pipe Properties node.

This example uses the Churchill relation (Ref. 2) that is valid for laminar flow, turbulent flow, and the transitional region in between these regimes. The Churchill relation is:

$$f_D = 8\left[\left(\frac{8}{\mathrm{Re}}\right)^{12} + (A+B)^{-1.5}\right]^{1/12} \tag{3}$$

where

$$A = \left[-2.457\ln\left(\left(\frac{7}{\mathrm{Re}}\right)^{0.9} + 0.27(e/d)\right)\right]^{16} \tag{4}$$

$$B = \left(\frac{37530}{\mathrm{Re}}\right)^{16} \tag{5}$$

As seen from the equations above, the friction factor is a function of the surface roughness divided by diameter of the pipe. Surface roughness data can be selected from a predefined list in the **Pipe Properties** feature.

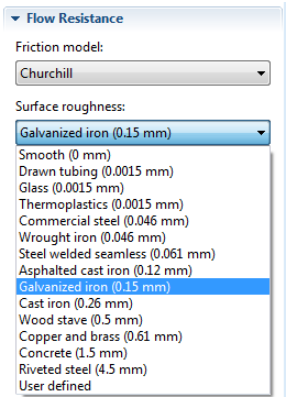

Figure 3: Specify the Pipe Shape and the Surface roughness in the Pipe Properties node.

The Churchill equation is also a function of the fluid properties and flow type, and geometry, through the Reynolds number:

$$\mathrm{Re} = \frac{\rho u d}{\mu} \tag{6}$$

The physical properties of water as function of temperature are directly available from the software's built-in material library.

Additional Flow Resistances

In pipe networks, fittings, bends, valves, and so on, induce additional lumped pressure losses

$$\Delta p = \frac{1}{2}\rho K_i u^2 \tag{7}$$

characterized by loss coefficients, K_i. The Pipe Flow interface can include such resistances through the following point features:

- 90° bend
- 45° bend
- T-junction
- Sudden contraction

- Gradual contraction
- Sudden expansion
- Gradual expansion
- Globe valve
- Gate valve
- Angle valve
- Ball valve
- Butterfly valve
- Swing check

This model uses two 90° bends and a Ball valve.

Results and Discussion

Figure 4 shows the pressure drop over the pipe system, while Figure 5 shows the direction of flow and the fluid velocity.

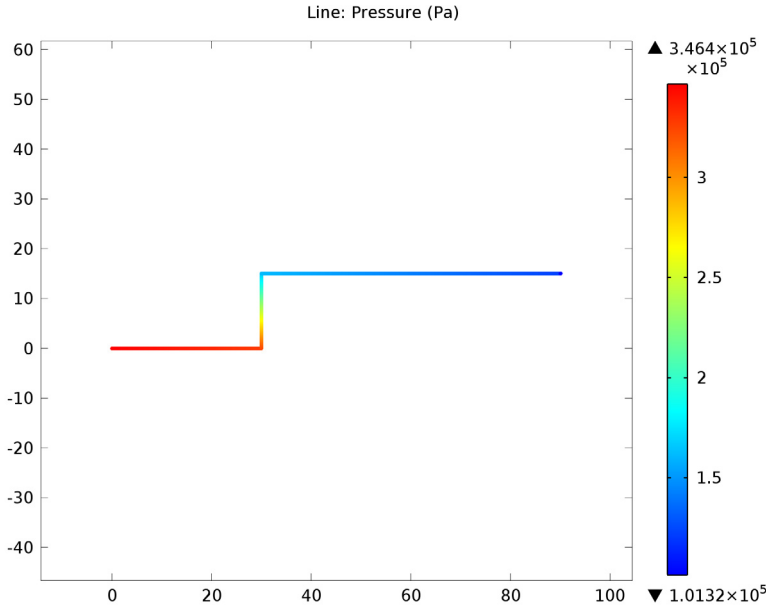

Figure 4: Pressure drop across the pipe system.

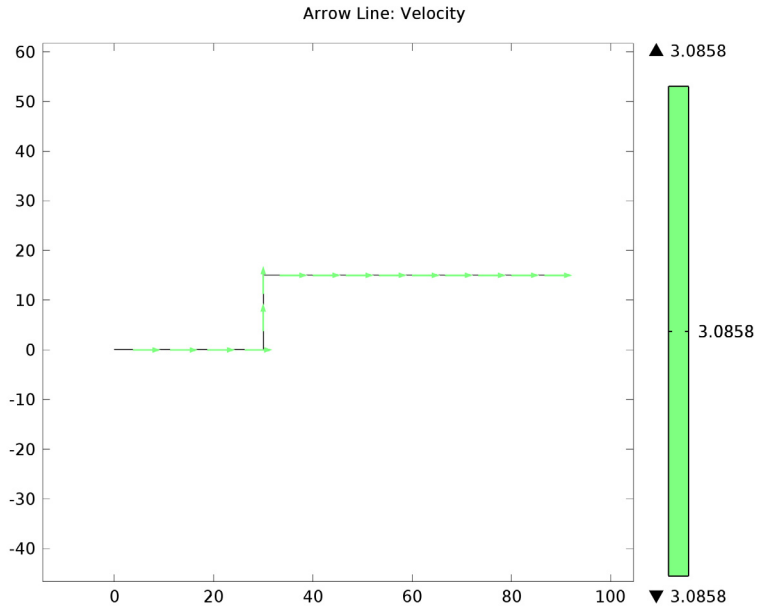

Figure 5: The fluid velocity is constant at approximately 3.1 m/s.

The initial discharge rate is calculated to 54.5 m^3/s. A check of the Reynolds number produces Re = $4.6 \cdot 10^5$ and indicates that the flow is well in the turbulent regime.

References

1. J.M. Coulson, J.F. Richardson, *Chemical Engineering vol. 1,* 4th ed., Pergamon Press, pp. 74–75, 1990.

2. S.W. Churchill, "Friction factor equations span all fluid-flow regimes," *Chem. Eng.*, vol. 84, no. 24, p. 91, 1997.

Model Library path: `Pipe_Flow_Module/Tutorial_Models/discharging_tank`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

1 In the **New** window, click the **Model Wizard** button.

MODEL WIZARD

1 In the **Model Wizard** window, click the **2D** button.

2 In the **Select physics** tree, select **Fluid Flow>Single-Phase Flow>Pipe Flow (pfl)**.

3 Click the **Add** button.

4 Click the **Study** button.

5 In the tree, select **Preset Studies>Stationary**.

6 Click the **Done** button.

GEOMETRY 1

Polygon 1 (pol1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Polygon**.

2 In the **Polygon** settings window, locate the **Coordinates** section.

3 In the **x** edit field, type `0 30 30 90 90.1`.

4 In the **y** edit field, type `0 0 15 15 15`.

5 Locate the **Object Type** section. From the **Type** list, choose **Open curve**.

MATERIALS

Now go to the Material Browser and select Water from the built-in Liquids and Gases library. The material properties will apply to the entire model domain by default.

On the **Home** toolbar, click **Add Material**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Water, liquid**.

3 In the **Add material** window, click **Add to Component**.

MATERIALS

Water, liquid (mat1)

1 Click the **Zoom Extents** button on the Graphics toolbar.

PIPE FLOW (PFL)

Next, specify the dimensions and surface roughness of the pipe. Note that you can add multiple Pipe Properties features and assign them to different parts of a pipe network, should you have a system of made up of pipes with different characteristics.

Pipe Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Pipe Flow (pfl)** click **Pipe Properties 1**.

2 In the **Pipe Properties** settings window, locate the **Pipe Shape** section.

3 From the list, choose **Round**.

4 In the d_i edit field, type `15[cm]`.

5 Locate the **Flow Resistance** section. From the **Surface roughness** list, choose **Galvanized iron (0.15 mm)**.

Pressure 2

Set boundary conditions for the inlet and outlet and use a Volume Force feature to take gravity effects into account.

1 On the **Physics** toolbar, click **Points** and choose **Pressure**.

2 Select Point 1 only.

3 In the **Pressure** settings window, locate the **Boundary Pressure** section.

4 In the p_0 edit field, type `101325[Pa]+(25[m])*g_const*pfl.rho`.

Volume Force 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Volume Force**.

2 Select Boundaries 1–3 only.

The default volume force is a gravity vector pointing in the negative y-direction.

3 Click the **Zoom Extents** button on the Graphics toolbar.

Bend 1

Next, add a number of point features to include the pressure losses due to bends and the ball valve. The valve point may be difficult to select graphically with the mouse. Here, you can use the Selection list, as help to browse the points in a list.

1 On the **Physics** toolbar, click **Points** and choose **Bend**.

2 Select Points 2 and 3 only.

Valve 1

1 On the **Physics** toolbar, click **Points** and choose **Valve**.

2 Select Point 4 only.

3 In the **Valve** settings window, locate the **Valve Specification** section.

4 From the **Valve** list, choose **Ball valve (K = 4.5)**.

MESH 1

Now, mesh and solve the model.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Extra fine**.

4 Click the **Build All** button.

5 Right-click **Component 1 (comp1)>Mesh 1** and choose **Statistics**.

STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Velocity (pfl)

Default plots show the pressure drop in the pipe system, and the direction and velocity of the flow. Now select from predefined plot quantities to evaluate the volumetric flow rate and the Reynolds number.

1 In the **Model Builder** window, expand the **Results>Velocity (pfl)>Arrow Line 1.1** node, then click **Color Expression 1.1**.

2 In the **Color Expression** settings window, locate the **Expression** section.

3 Click the Replace expression button in the upper-right corner of the section and from the menu choose **Pipe Flow> Mass flow rate> Mass flow rate magnitude (pfl.Qm).**

4 Click the **Replace expression** button in the upper-right corner of the section and and from the menu choose **Pipe Flow> Reynolds number (pfl.Re).**