Created in COMSOL Multiphysics 5.6

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 energy 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.

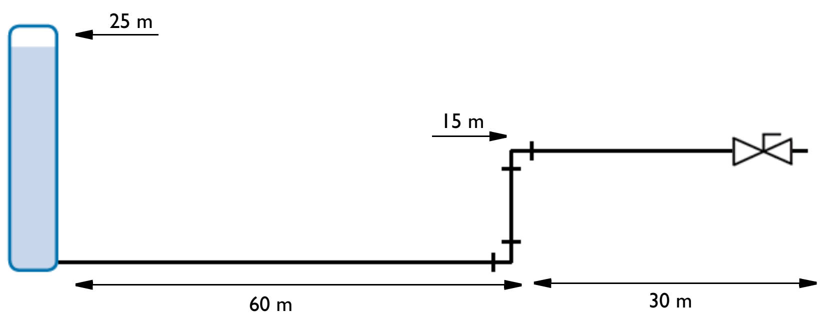

Figure 1: Water flows through a pipe system with two 90° bends and discharges through an open ball valve.

The model example is taken from Ref. 1.

THE MOMENTUM EQUATION

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

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

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: $\mathrm{N/m^3}$), in this case used to account for gravity. ρ and A in Equation 2 are fluid density ($\mathrm{kg/m^3}$) and cross section area ($\mathrm{m^3}$), 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 User'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.

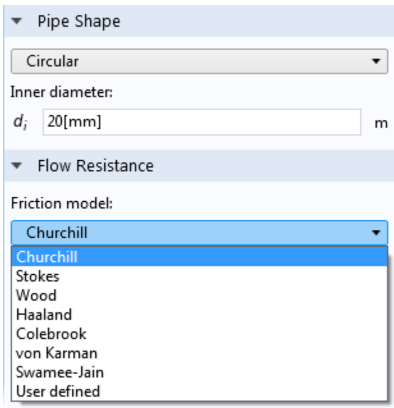

Figure 2: Select from different predefined Friction models in the Pipe Properties node.

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:

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

where

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

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

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.

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

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

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 energy losses

$$\Delta e = \frac{1}{2}K_i u^2$$

characterized by loss coefficients, K_i. The Pipe Flow interface can include such resistances through the point features. This model uses two 90° bends and a Ball valve.

BOUNDARY CONDITIONS

At the pipe inlet from the tank, see Figure 1, the pressure is taken as the atmospheric pressure at the top water surface in the tank plus the hydrostatic pressure due to the water column:

$$p_{\mathrm{in}} = p_0 + \rho g h$$

where g is the normal gravitational acceleration ($\mathrm{m/s^2}$) and the h the elevation height (m), the latter which is 25 m in this case. At the system outlet to the right, atmospheric pressure p_0 is specified.

Results and Discussion

Figure 3 shows the pressure drop over the pipe system, while Figure 4 shows the direction of flow and the fluid velocity.

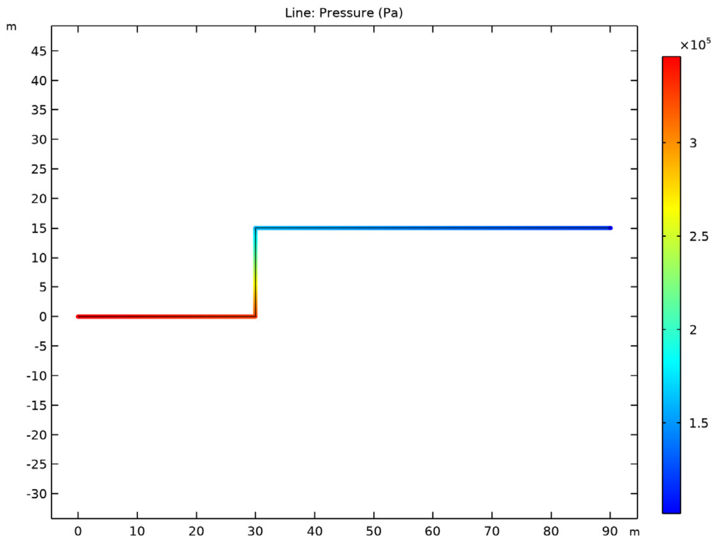

Figure 3: Pressure drop across the pipe system.

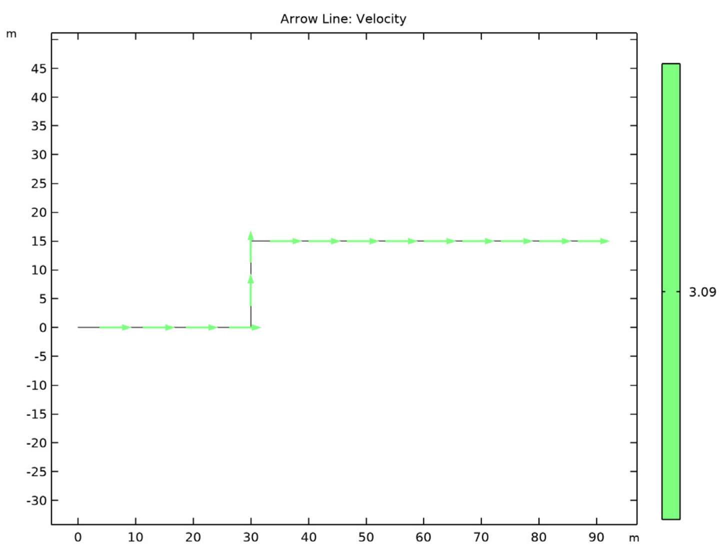

Figure 4: The fluid velocity is constant at approximately 3.1 m/s.

The initial discharge rate is calculated to 54.5 m^3/s. A check of the Reynolds number produces Re = $4.58 \cdot 10^5$ and indicates that the flow is well in the turbulent regime.

References

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

2. S.W. Churchill, “Friction factor equation spans all fluid-flow regimes,” *Chem. Eng.*, vol. 84, no. 24, p. 91, 1997.

Application Library path: `Pipe_Flow_Module/Tutorials/discharging_tank`

Modeling Instructions

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

NEW

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

MODEL WIZARD

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

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

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. In the table, enter the following settings:

x (m)	y (m)
0	0
30	0
30	15
90	15
90.1	15

MATERIALS

Now add Water from the Material Library. The material properties will apply to the entire model domain by default.

ADD MATERIAL

1. In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **Built-in>Water, liquid**.
4. Click **Add to Component** in the window toolbar.
5. In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Water, liquid (mat1)

Click the **Zoom Extents** button in 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 **Settings** window for **Pipe Properties**, locate the **Pipe Shape** section.
3. From the list, choose **Circular**.
4. In the d_i text field, type `15[cm]`.
5. Locate the **Flow Resistance** section. From the **Surface roughness** list, choose **Galvanized iron (0.15 mm)**.

Pressure 2

1. In the **Physics** toolbar, click **Points** and choose **Pressure**.
2. Select Point 1 only.
3. In the **Settings** window for **Pressure**, locate the **Boundary Pressure** section.
4. In the p_0 text field, type `101325[Pa]+(25[m])*g_const*pfl.rho`.

Volume Force 1

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

1. In 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 (downward).
3. Click the **Zoom Extents** button in the **Graphics** toolbar.

Next, add a number of point features to include the energy 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.

Bend 1

1. In the **Physics** toolbar, click **Points** and choose **Bend**.
2. Select Points 2 and 3 only.

Valve 1

1. In the **Physics** toolbar, click **Points** and choose **Valve**.
2. Select Point 4 only.
3. In the **Settings** window for **Valve**, locate the **Valve Specification** section.
4. From the **Valve** list, choose **Ball valve (K = 4.5)**.

   To help graphically indicate locations of pipe network elements, the physics symbols are available.
5. In the **Model Builder** window, click **Pipe Flow (pfl)**.
6. In the **Settings** window for **Pipe Flow**, locate the **Physics Symbols** section.
7. Select the **Enable physics symbols** check box.
8. Find the **Show or hide all physics symbols** subsection. Click **Select All** to display physics symbols for all features.

STUDY 1

The model is now ready for solving. Lower the default tolerance to increase the accuracy of the solution.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Stationary Solver 1**.

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1e-6`.

5 In the **Study** toolbar, click **Compute**.

RESULTS

Velocity (pfl)

Default plots show the pressure drop in the pipe system, and the direction and velocity of the flow (Figure 3 and Figure 4). Now select from predefined plot quantities to evaluate the volumetric flow rate and the Reynolds number.

1 In the **Model Builder** window, expand the **Velocity (pfl)** node.

Color Expression 1

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

2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Pipe Flow>pfl.Re - Reynolds number**.

3 Click to expand the **Title** section. From the **Title type** list, choose **Automatic**.

4 In the **Velocity (pfl)** toolbar, click **Plot**.