



Model created in COMSOL Multiphysics 6.4

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.

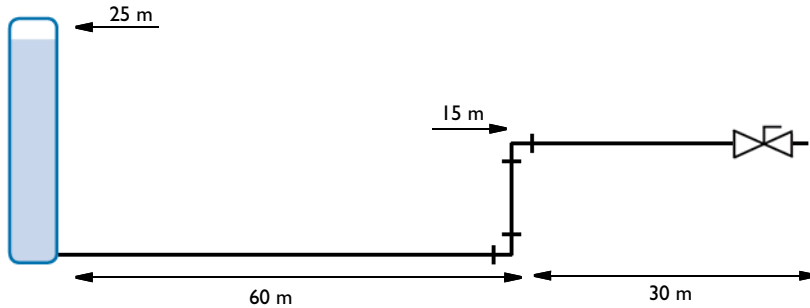


Figure 1: Water flows through a pipe system with two 90° bends and discharges through an open ball valve.

The model example is taken from [Ref. 1](#).

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

$$\Delta p_{\text{tot}} = \frac{1}{2} K u^2$$

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

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:

$$p_{\text{res}} = p_0 + \rho g H$$

where g is the normal gravitational acceleration (m/s^2) and the H the elevation height (m), the latter which is 25 m in this case. The entrance from the tank is rounded. At the system outlet to the right, atmospheric pressure p_0 is specified. The pressure drop across the pipe system is computed for a constant water level $H=H_0$.

Results and Discussion

Figure 2 shows the pressure drop over the pipe system for stationary solution, while Figure 3 shows the direction of flow and the Reynolds number.

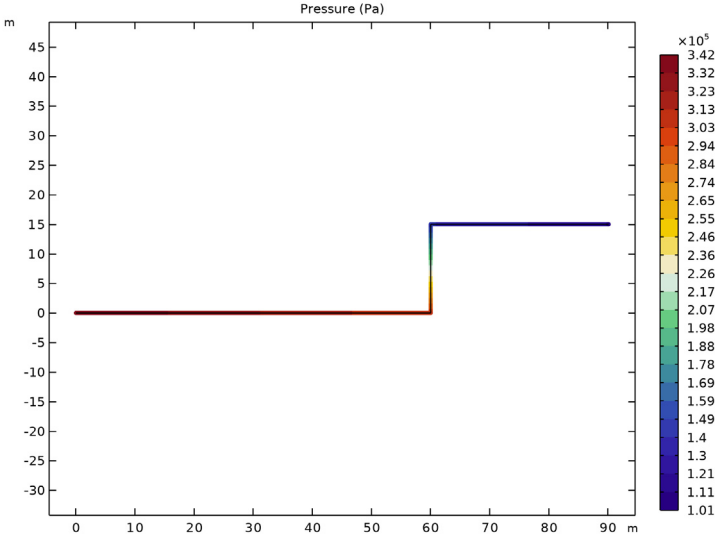


Figure 2: Pressure drop across the pipe system.

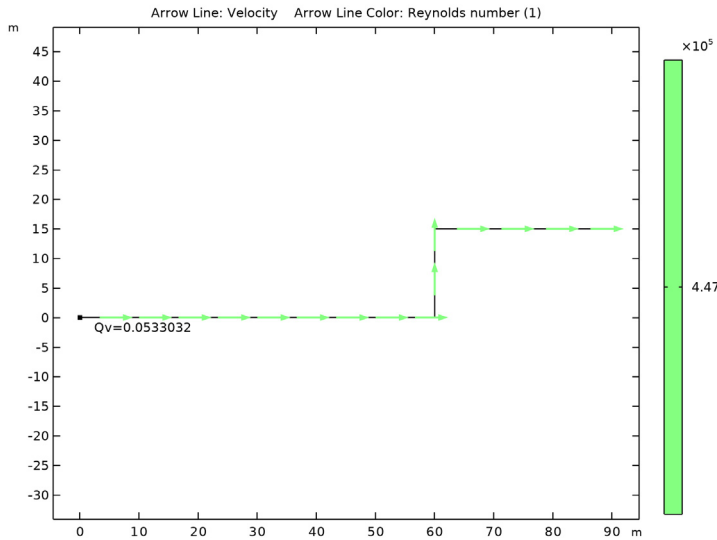


Figure 3: The Reynolds number is $4.47 \cdot 10^5$, indicating that the flow is well in the turbulent regime.

The discharge rate is calculated to approximately $0.053 \text{ m}^3/\text{s}$.

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**.


GLOBAL DEFINITIONS

Parameters I

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 Click  **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `discharging_tank_parameters.txt`.

GEOMETRY I

Polygon I (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
L1	0
L1	L2
V1	L2
L1+L3	L2

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 **Materials** 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 the **Add to Component** button in the window toolbar.
- 5 In the **Materials** 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 Dh.
- 5 Locate the **Flow Resistance** section. From the **Surface roughness** list, choose **Galvanized iron (0.15 mm)**.


Inlet 1

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


- 1 In the **Physics** toolbar, click  **Points** and choose **Inlet**.
- 2 In the **Settings** window for **Inlet**, locate the **Inlet Specification** section.
- 3 From the **Specification** list, choose **Reservoir**.
- 4 In the p_{res} text field, type $101325[\text{Pa}]+H0*\text{g_const}*pfl.\text{rho}$.

- 5 Select Point 1 only.
- 6 From the **Entrance type** list, choose **Rounded (K = 0.05)**.

Volume Force 1


- 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** checkbox.
- 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.

Step 1: Stationary

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Study Settings** section.

- 3 From the **Tolerance** list, choose **User controlled**.
- 4 In the **Relative tolerance** text field, type 1e-6.
- 5 In the **Study** toolbar, click  **Compute**.

RESULTS

Pressure (pfl)

Default plots show the pressure drop in the pipe system, and the direction and velocity of the flow (Figure 2 and Figure 3). Modify the last plot to evaluate the Reynolds number and flow rate.





Velocity (pfl)

In the **Model Builder** window, expand the **Results > 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 - 1**.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **Automatic**.

Annotation 1

- 1 In the **Model Builder** window, right-click **Velocity (pfl)** and choose **Annotation**.
- 2 In the **Settings** window for **Annotation**, locate the **Annotation** section.
- 3 Click  **Replace Expression**.
- 4 Click the  button. From the menu, choose **Component 1 (comp1) > Pipe Flow > pfl.Qv - Volumetric flow rate magnitude - m³/s**.
- 5 In the **Text** text field, type $Qv=eval(pfl.Qv)$.
- 6 In the **Velocity (pfl)** toolbar, click  **Plot**.
- 7 Click the  **Zoom Extents** button in the **Graphics** toolbar.