

Fluid Valve

Introduction

Many different applications such as printers involve the periodic opening and closing of fluid-flow channels. This is generally a difficult problem to model as it implies a moving boundary condition for the part of the geometry that acts as an obstacle for the flow.

An alternative is to use a material property (viscosity), which is easy to vary over time. In this case, specify a very large viscosity (ideally it should be an infinite viscosity) that in effect stops the flow in regions where this large viscosity is present. To simulate the movement of this region of large viscosity, the model includes a logical expression in the subdomain settings.

The model describes a valve where it is possible to direct the flow into one of the two channels. Flow of varying degrees can also occur in both channels during the opening and closing stages.

Model Definition

Figure 1 shows the model domain.

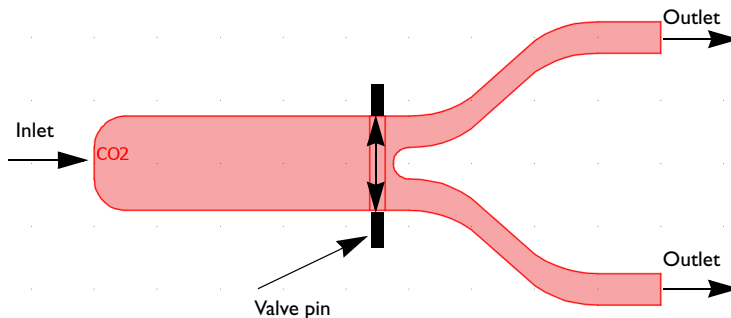


Figure 1: Depiction of the geometry and the operation of the fluid valve. Flow enters from one inlet at the left, but can leave the valve through two outlets at the right. The choice of outlet depends upon the position of the valve pin. In this model, two valve pins oscillate between the following positions: in front of one outlet channel to in front of the other. Sometimes flow is possible through both outlets, depending on the position of the pins.

In this model, the valve pin moves according to a sinusoidal function of time. The inlet velocity is constant.

DOMAIN EQUATIONS

The fluid flow is described by the Navier-Stokes equations:

$$\begin{aligned} \rho \frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) + \rho \mathbf{u} \cdot \nabla \mathbf{u} + \nabla p &= 0 \\ \nabla \cdot \mathbf{u} &= 0 \end{aligned} \quad (1)$$

where ρ denotes the density (kg/m^3), \mathbf{u} the velocity vector (m/s), η the viscosity (Ns/m^2), and p the pressure (Pa). The modeled fluid is air with viscosity $10^{-5} \text{Ns}/\text{m}^2$ and density $1 \text{kg}/\text{m}^3$.

The movement of the valve is described with an analytic expression pin , which returns the value of one in the area corresponding to the valve pin and zero elsewhere. The viscosity is then expressed by

$$\eta = \eta_0 + \text{pin} \cdot \eta_\infty \quad (2)$$

where η_0 is the fluid viscosity, η_∞ is a very large viscosity (ideally infinite), and pin is described by

$$\text{pin} = x_{\text{pin}} y_{\text{pin}} \quad (3)$$

where

$$x_{\text{pin}} = (x > x_0)(x < x_1) \quad (4)$$

$$y_{\text{pin}} = 1 - (y > y_1) + (y > y_2) \quad (5)$$

y_1 and y_2 depend on time, t , according to:

$$y_1 = -y_0 + y_{\text{max}} \sin(2\pi t) \quad (6)$$

$$y_2 = y_0 + y_{\text{max}} \sin(2\pi t)$$

and where y_0 , x_0 , x_1 , and y_{max} are fixed in time and describe the size of the valve pin and the amplitude with which the pin moves.

BOUNDARY CONDITIONS

At the inlet, the model uses a fully developed laminar flow. The velocity is set to a parabolic velocity profile with maximum velocity v_{max} equal to $0.25 \text{m}/\text{s}$. At the outlets, a neutral boundary condition states that the normal component of the stress tensor is zero:

$$\mathbf{n} \cdot [-p\mathbf{I} + \eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)] = 0 \tag{7}$$

All other boundaries have a no-slip condition

$$\mathbf{u} = 0 \tag{8}$$

Results

Figure 2 shows the velocity field (modulus of the velocity vector) when the valve is completely open. Firstly, the plot shows that the inlet is smaller than the compartment that it enters, so that some distance is required before the flow reaches another parabolic profile for the main body of the inlet chamber. If you investigate the velocity, some recirculation occurring at the corners of the chamber beside the inlet is visible.

Secondly, the structure of the outlet channels leads to a slight thinning toward the middle of the channels. This provides a slight acceleration, and subsequently greater velocity magnitude, in these regions.

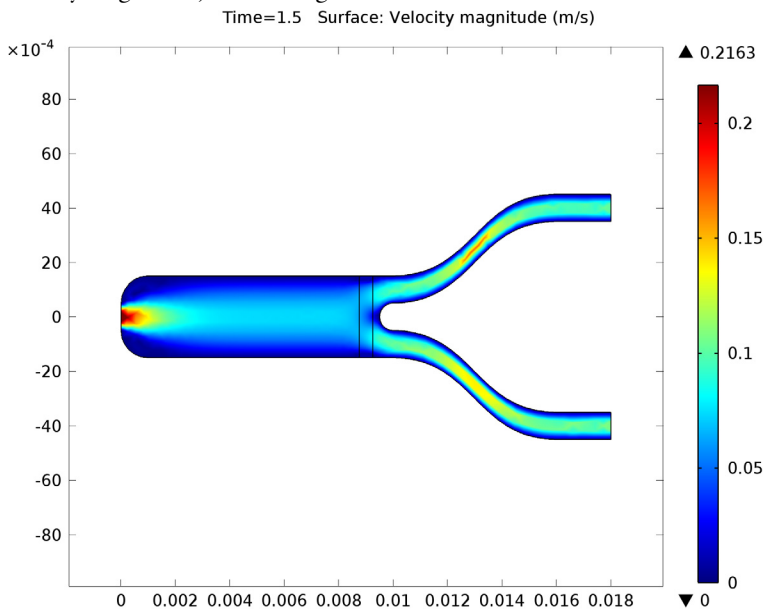


Figure 2: Velocity field when the valve is completely open.

Figure 3 shows the velocity field when the valve is halfway closed from above and fully closed from below. The use of a very large viscosity not only simulates the solidity of

the valve pin but also does a good job of providing a no-slip boundary condition at the edge of the valve pin.

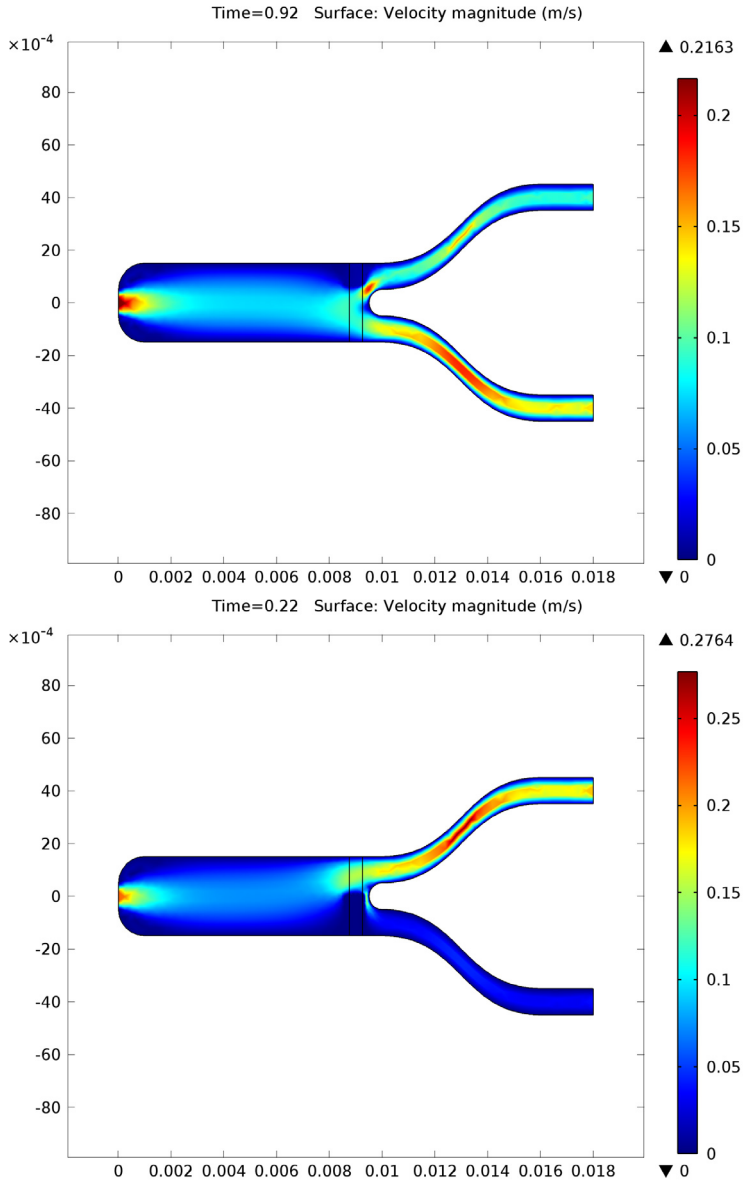


Figure 3: Velocity field when the valve is half way closed from above (top) and fully closed from below (bottom).

Figure 4 shows a plot of the flow rates (integral of the velocity vector over the outlets) in the upper and lower channels as well as the total flow rate. The figure illustrates the periodic flow due to the periodic motion of the valve pin.

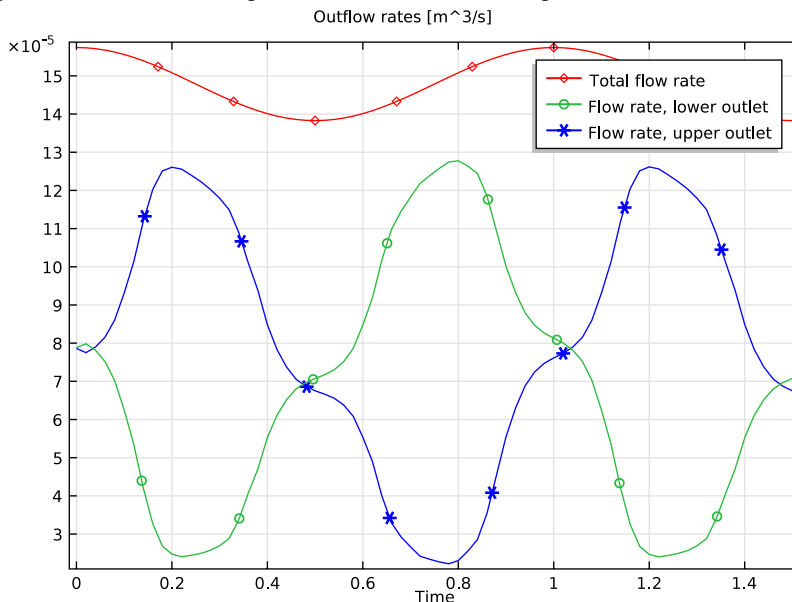


Figure 4: The blue line (solid) shows flow rate in the upper branch as a function of time, while the green line shows the flow rate in the lower branch. The red line shows the sum of the flow in the two branches.

Notes About the COMSOL Implementation

The inequalities used to describe the pin motion are, in the COMSOL Multiphysics implementation, replaced by smooth step functions.

Model Library path: COMSOL_Multiphysics/Fluid_Dynamics/fluid_valve

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

- 2 Click the **2D** button.
- 3 Click **Next**.
- 4 In the **Add physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.
- 5 Click **Next**.
- 6 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.
- 7 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 Go to the **Settings** window for Parameters.
- 3 Locate the **Parameters** section. In the **Parameters** table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
v_max	0.25[m/s]	Maximum inlet velocity
eta_0	1e-5[Pa*s]	Fluid viscosity
eta_inf	1[Pa*s]	Large viscosity
x0	8.7[mm]	Valve, left end point
x1	9.2[mm]	Valve, right end point
y0	1.5[mm]	Pin height parameter
y_max	1.5[mm]	Pin amplitude
scale	1e-4	Step function scaling

DEFINITIONS

Variables 1

- 1 In the **Model Builder** window, right-click **Model 1>Definitions** and choose **Variables**.
- 2 Go to the **Settings** window for Variables.

3 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
v_pin	$y_{\max} \pi^2 [1/s] \cos(2\pi t [1/s])$	Pin velocity
y1	$-y_0 + y_{\max} \sin(2\pi t [1/s])$	Lower pin top position
y2	$y_0 + y_{\max} \sin(2\pi t [1/s])$	Upper pin bottom position
x_range	$\text{step1}((x-x_0)[1/m]) * (1 - \text{step1}((x-x_1)[1/m]))$	Horizontal pin range
y_range	$1 - \text{step1}((y-y_1)[1/m]) + \text{step1}((y-y_2)[1/m])$	Vertical pin range
pin	$y_{\text{range}} * x_{\text{range}}$	Pin extension expression
eta	$\text{eta}_0 + \text{eta}_{\text{inf}} * \text{pin}$	Viscosity expression

Note that the unit syntax used in step function calls for x_range and y_range is optional.

Now define a step function using the scale parameter you defined in the Parameters node.

Step 1

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Functions>Step**.
- 2 Go to the **Settings** window for Step.
- 3 Click to expand the **Smoothing** section.
- 4 In the **Size of transition zone** edit field, type $2 * \text{scale}$.

GEOMETRY 1

Import 1

- 1 In the **Model Builder** window, right-click **Model 1>Geometry 1** and choose **Import**.
- 2 Go to the **Settings** window for Import.
- 3 Locate the **Import** section. Click the **Browse** button.
- 4 Browse to the model's Model Library folder and double-click the file fluid_valve.mphbin.
- 5 Click the **Import** button.

MATERIALS

Material 1

- 1 In the **Model Builder** window, right-click **Model 1>Materials** and choose **Material**.

- 2 Go to the **Settings** window for Material.
- 3 Locate the **Material Contents** section. In the **Material contents** table, enter the following settings:

PROPERTY	NAME	VALUE
Density	rho	1 [kg/m ³]
Dynamic viscosity	mu	eta

LAMINAR FLOW

The required fluid properties, density and viscosity are defined by the material. Thus, you can directly proceed with defining the boundary conditions.

Inlet 1

- 1 In the **Model Builder** window, right-click **Model 1 > Laminar Flow** and choose **Inlet**.
- 2 Go to the **Settings** window for Inlet.
- 3 Locate the **Velocity** section. In the U_0 edit field, type $4*v_max*s*(1-s)$.
- 4 Select Boundary 1 only.

Inlet 2

- 1 In the **Model Builder** window, right-click **Laminar Flow** and choose **Inlet**.
- 2 Select Boundaries 5 and 6 only.
- 3 Go to the **Settings** window for Inlet.
- 4 Locate the **Velocity** section. In the U_0 edit field, type v_pin .

Open Boundary 1

- 1 In the **Model Builder** window, right-click **Laminar Flow** and choose **Open Boundary**.
- 2 Select Boundaries 14 and 15 only.

DEFINITIONS

Next, define two integration coupling operators on the two outlet boundaries. You will use them to calculate flow rates in the upper and lower channels.

Integration 1

- 1 In the **Model Builder** window, right-click **Model 1 > Definitions** and choose **Model Couplings > Integration**.
- 2 Go to the **Settings** window for Integration.
- 3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

- 4 Select Boundary 15 only.

Integration 2

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.
- 2 Go to the **Settings** window for Integration.
- 3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 14 only.

Variables 1

Add two new variables that use the just defined integration coupling operators.

- 1 In the **Model Builder** window, click **Variables 1**.
- 2 Go to the **Settings** window for Variables.
- 3 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION
u_up	intop1(u)
u_down	intop2(u)

MESH 1

Free Triangular 1

- 1 In the **Model Builder** window, right-click **Model 1>Mesh 1** and choose **Free Triangular**.
- 2 Go to the **Settings** window for Free Triangular.
- 3 Locate the **Domain Selection** section. From the **Geometric entity level** list, choose **Entire geometry**.

Specify a finer mesh on the pin domain.

Size 1

- 1 Right-click **Free Triangular 1** and choose **Size**.
- 2 Go to the **Settings** window for Size.
- 3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 2 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.

- 6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 7 In the associated edit field, type 0.1 [mm].

Now, define the size for the remaining domains in the default size node.

Size

- 1 In the **Model Builder** window, click **Size**.
- 2 Go to the **Settings** window for Size.
- 3 Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type 0.4 [mm].
- 4 In the **Maximum element growth rate** edit field, type 1.5.
- 5 Click the **Build All** button.

STUDY I

Specify the time range, then compute the solution.

Step 1: Time Dependent

- 1 In the **Model Builder** window, expand the **Study I** node, then click **Step 1: Time Dependent**.
- 2 Go to the **Settings** window for Time Dependent.
- 3 Locate the **Study Settings** section. In the **Times** edit field, type range (0, 0.02, 1.5).
- 4 In the **Model Builder** window, right-click **Study I** and choose **Compute**.

RESULTS

Velocity (spf)

Plot the velocity magnitude at different time steps.

- 1 In the **Model Builder** window, click **Results>Velocity (spf)**.
- 2 Go to the **Settings** window for 2D Plot Group.
- 3 Locate the **Data** section. From the **Time** list, choose **0.22**. Click the **Plot** button.
- 4 Repeat the previous instruction for the times 0.22 s and 0.92 s. to generate [Figure 3](#).

ID Plot Group 3

Finally, plot of the flow rates in the upper and lower channels as well as the total flow rate ([Figure 4](#)).

- 1 In the **Model Builder** window, right-click **Results** and choose **ID Plot Group**.
- 2 Right-click **Results>ID Plot Group 3** and choose **Global**.
- 3 Go to the **Settings** window for Global.
- 4 Locate the **y-Axis Data** section. In the table, enter the following settings:

EXPRESSION	DESCRIPTION
u_up	Flow rate, upper outlet
u_down	Flow rate, lower outlet
u_up+u_down	Total flow rate

- 5 Click to expand the **Coloring and Style** section.
- 6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 7 Locate the **Legends** section. Select the **Show legends** check box.
- 8 In the **Model Builder** window, click **ID Plot Group 3**.
- 9 Go to the **Settings** window for ID Plot Group.
- 10 Locate the **Title** section. From the **Title type** list, choose **Manual**.
- 11 In the **Title** text area, type Outflow rates [m³/s].
- 12 Click the **Plot** button.

The figure illustrates the periodic flow due to the periodic motion of the valve pin.

