Created in COMSOL Multiphysics 6.0

Star-Shaped Microchannel

Introduction

Lab-on-a-chip devices have become quite popular for analyses in fields such as biochemistry and bioengineering. Through various techniques they incorporate all the equipment involved in a chemical process such as chemical reactors, heat exchangers, separators, and mixers.

This example involves the design of an infuser, a device that feeds a reactor or analysis equipment with a specific amount of fluid. Controlling pressure is an accurate way to introduce a set quantity of fluid at a certain velocity to some piece of equipment.

Flushing the equipment can also be important. Optimizing such an infuser to maximize its use would involve spending the least amount of time (and fluid) flushing the equipment. Modeling this process in the time domain can lead to an optimization of the infusing pressure, microchannel design, and time control.

This model demonstrates two useful tools in COMSOL Multiphysics modeling:

- The ability to easily define a time-dependent boundary condition
- The ability to easily sweep meshes into 3D to save memory

Model Definition

This exercise arbitrarily sets the geometry and conditions of the microchannel lab-on-a-chip (Figure 1). The differential pressure at the five inlets relative to the outlet pressure is time-controlled so that the inlet flow passes from one to the next in a smooth way. At any particular instant, one of the inlet flows dominates, although flow could be significant from more than one inlet. The pressure at the outlet is set to zero.

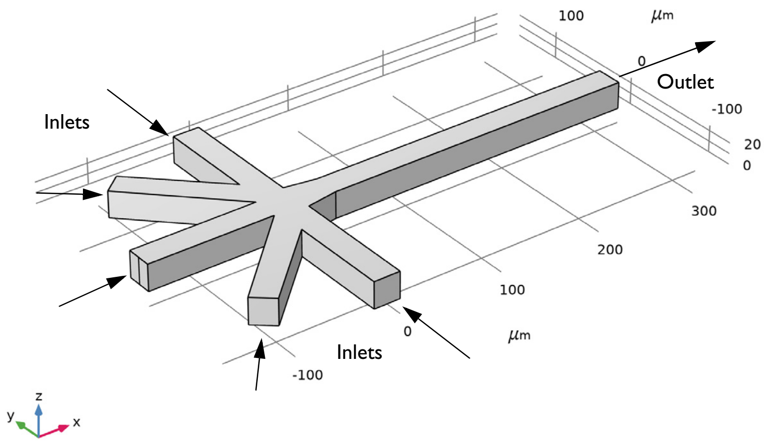

Figure 1: Model geometry for a star-shaped infuser with five inlets and one outlet. The model sets up a varying pressure differential at each inlet in the time domain in such a way that the dominant inlet flow alternates among them.

The example models only fluid flow whose velocity is of a magnitude that suggests laminar behavior. This implies that you can get a numerical solution of the full momentum balance and continuity equations for incompressible flow with a reasonable number of elements. The equations you must solve are the Navier-Stokes equations in the time domain

$$\rho\frac{\partial \mathbf{u}}{\partial t} - \nabla\cdot\mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) + \rho\mathbf{u}\cdot\nabla\mathbf{u} + \nabla p = 0$$

$$\nabla\cdot\mathbf{u} = 0$$

where ρ denotes density ($\mathrm{kg/m^3}$), $\mathbf{u}$ is the velocity (m/s), μ denotes dynamic viscosity (Pa·s), and p equals pressure (Pa). The fluid in this case is water, with the corresponding density and viscosity values.

The boundary conditions for the inlets and the outlet assume a set pressure; they also assume vanishing viscous stress:

$$[\nabla\cdot\mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)]\cdot\mathbf{n} = 0$$

$$p = p_{\mathrm{i}}$$

Set the pressure at the outlet to zero; at the inlets, use the time-dependent expressions

$$p_{\mathrm{i}} = 50 + 10\sin(\pi t + \alpha) \quad [\mathrm{Pa}]$$

where t is time (s). This simplified example sets the phase α to 0, $\pi/4$, $\pi/2$, $3\pi/4$, or π, depending on the inlet boundary.

Apply the no slip condition to all other boundaries; it states that the velocity is zero in the x, y, and z directions at the wall:

$$\mathbf{u} = (0, 0, 0)$$

Results

Figure 2 shows the velocity field as a combined slice and arrow plot through the middle of the geometry at $t = 0.5$ s. Setting up and observing this plot as an animation gives an informative qualitative description of the process.

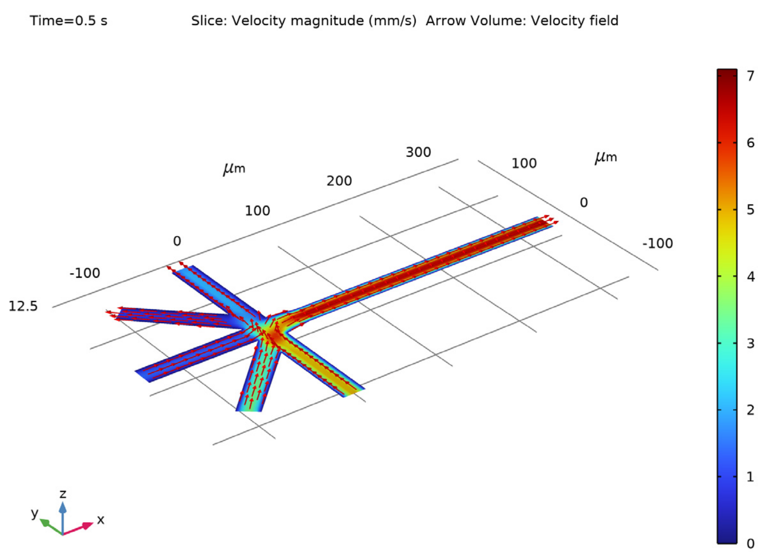

Figure 2: The velocity field in a microchannel infuser through the middle of the geometry.

Figure 3 shows the velocity in the x direction and the pressure in a point near the outlet as functions of time.

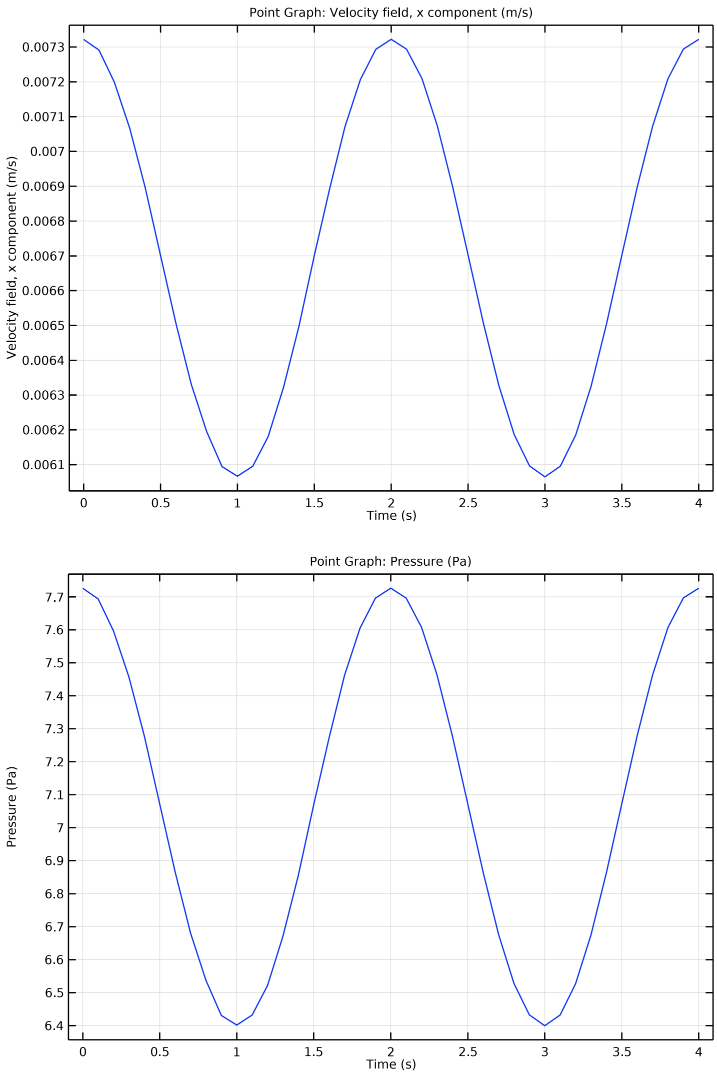

Figure 3: Velocity in the x direction (top) and pressure (bottom) at a point near the outlet.

Notes About the COMSOL Implementation

This example illustrates how to use time-dependent boundary conditions to simulate a changing process. You can implement this scenario using a boundary condition that is a function of time. The physics interface provides direct access to the built-in time variable (t) and the mathematical functions you need.

In 3D models, results at the walls are important but they can also hide what occurs within the geometry. This example also illustrates how to better display results with the help of hidden boundaries.

Finally, this model approaches meshing in a way that deviates from the default settings. In most cases COMSOL Multiphysics automatically generates a 3D mesh made completely of tetrahedra. Here — as is the case in many other microchannels and minichannels — the top and the bottom boundaries are significant in modeling the flow profile because the distance between them is of the same magnitude as that between the two sides. This means that you must model the device in 3D. However, because the microchannel's height does not change along its length, the software does not require much meshing to resolve this dimension.

As an alternative to its default meshing, it is possible to extrude a mesh. To illustrate this concept, you create the mesh in this model by first taking a cross section of the full geometry to construct a 2D geometry. After meshing that, you then extrude the geometry and sweep the mesh in the height dimension. This approach provides some mesh and memory conservation.

Application Library path: `Microfluidics_Module/Fluid_Flow/star_chip`

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 **3D**.

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

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **µm**.

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Polygon 1 (pol1)

1 In the **Work Plane** toolbar, click **Polygon**.

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

3 From the **Data source** list, choose **Vectors**.

4 In the **xw** text field, type `325 325 30 0 0 -25 -25 -97 -115 -30 -150  -150 -30`.

5 In the **yw** text field, type `12.5 25 25 30 150 150 55 125 110 25 25 12.5 12.5`.

Work Plane 1 (wp1)>Mirror 1 (mir1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Mirror**.

2 Select the object **pol1** only.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Point on Line of Reflection** section. In the **xw** text field, type `82.5`.

6 In the **yw** text field, type `12.5`.

7 Locate the **Normal Vector to Line of Reflection** section. In the **xw** text field, type `0`.

8 In the **yw** text field, type `1`.

Work Plane 1 (wp1)>Union 1 (uni1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

Extrude 1 (ext1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (µm)
25

4 Click **Build All Objects**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
p0	50[Pa]	50 Pa	Pressure offset
p1	10[Pa]	10 Pa	Pressure amplitude
omega	pi[rad/s]	3.1416 rad/s	Angular velocity
t	0	0	Dummy variable for stationary study

DEFINITIONS

Variables 1

1 In the **Home** toolbar, click a= **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`p_in_rm`	`p0+p1*sin(omega*t)`		Pressure, rightmost inlet
`p_in_ir`	`p0+p1*sin(omega*t+pi/4)`		Pressure, inner-right inlet
`p_in_c`	`p0+p1*sin(omega*t+pi/2)`		Pressure, central inlet
`p_in_il`	`p0+p1*sin(omega*t+3*pi/4)`		Pressure, inner-left inlet
`p_in_lm`	`p0+p1*sin(omega*t+pi)`		Pressure, leftmost inlet

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1000	kg/m³	Basic
Dynamic viscosity	mu	1e-3	Pa·s	Basic

LAMINAR FLOW (SPF)

Inlet 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Inlet**.

2 Select Boundaries 1 and 5 only.

3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

4 From the list, choose **Pressure**.

5 Locate the **Pressure Conditions** section. In the p_0 text field, type `p_in_c`.

Inlet 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

4 From the list, choose **Pressure**.

5 Locate the **Pressure Conditions** section. In the p_0 text field, type `p_in_ir`.

Inlet 3

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 9 only.

3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

4 From the list, choose **Pressure**.

5 Locate the **Pressure Conditions** section. In the p_0 text field, type `p_in_il`.

Inlet 4

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 14 only.

3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

4 From the list, choose **Pressure**.

5 Locate the **Pressure Conditions** section. In the p_0 text field, type `p_in_rm`.

Inlet 5

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 16 only.

3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

4 From the list, choose **Pressure**.

5 Locate the **Pressure Conditions** section. In the p_0 text field, type `p_in_lm`.

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundaries 23 and 24 only.

MESH 1

Extruding a 2D mesh into 3D creates a more structured mesh than the default.

Swept 1

In the **Mesh** toolbar, click **Swept**.

Size

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Calibrate for** list, choose **Fluid dynamics**.

Distribution 1

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 In the **Number of elements** text field, type `8`.

4 Select Domain 1 only.

5 Click **Build All**.

STUDY 1

Time Dependent

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,0.1,4)`.

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

RESULTS

Velocity (spf)

Next, reproduce the plot in Figure 2, showing the velocity field at $t = 0.5$ s as a combined slice and arrow plot.

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

2 From the **Time (s)** list, choose **0.5**.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

This setting gives an unobstructed view of the slice you will specify, which shows the shape of the microchannel chip.

Slice

1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Slice**.

2 In the **Settings** window for **Slice**, locate the **Expression** section.

3 From the **Unit** list, choose **mm/s**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

5 In the **Planes** text field, type `1`.

6 In the **Velocity (spf)** toolbar, click **Plot**.

Arrow Volume 1

1. In the **Model Builder** window, right-click **Velocity (spf)** and choose **Arrow Volume**.
2. In the **Settings** window for **Arrow Volume**, locate the **Arrow Positioning** section.
3. Find the **x grid points** subsection. In the **Points** text field, type `35`.
4. Find the **y grid points** subsection. In the **Points** text field, type `35`.
5. Find the **z grid points** subsection. In the **Points** text field, type `1`.
6. Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Normalized**.
7. Select the **Scale factor** check box.
8. In the associated text field, type `2600`.
9. In the **Velocity (spf)** toolbar, click **Plot**.

Pressure (spf)

The second default plot shows the surface of the geometry with the pressure as a contour plot.

1. In the **Model Builder** window, under **Results** click **Pressure (spf)**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Time (s)** list, choose **0.5**.
4. In the **Pressure (spf)** toolbar, click **Plot**.

   To create the plots in Figure 3 showing the velocity in the *x* direction and the pressure at a point near the outlet, perform the following steps:

Cut Point 3D 1

1. In the **Results** toolbar, click **Cut Point 3D**.
2. In the **Settings** window for **Cut Point 3D**, locate the **Point Data** section.
3. In the **x** text field, type `275`.
4. In the **y** text field, type `12.5`.
5. In the **z** text field, type `12.5`.

Velocity x-direction

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Velocity x-direction` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D 1**.

Point Graph 1

1 Right-click **Velocity x-direction** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Point 3D 1**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Laminar Flow>Velocity and pressure>Velocity field - m/s>u - Velocity field, x component**.

5 In the **Velocity x-direction** toolbar, click **Plot**.

Outlet pressure

1 In the **Model Builder** window, right-click **Velocity x-direction** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Outlet pressure` in the **Label** text field.

Outlet pressure

1 In the **Model Builder** window, expand the **Outlet pressure** node, then click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, type `Outlet pressure` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Laminar Flow>Velocity and pressure>p - Pressure - Pa**.

4 In the **Outlet pressure** toolbar, click **Plot**.