Star-Shaped Microchannel Chip

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.

©2012 COMSOL

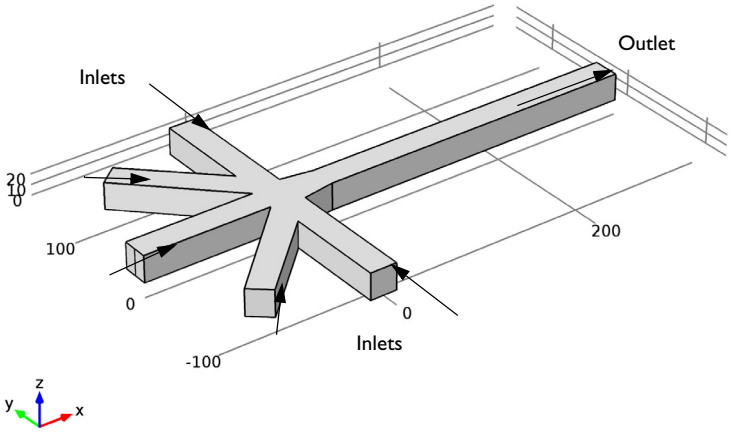

Figure 1: Model geometry for a star-shaped infuser with five inlets and one outlet. The model sets up a varying pressure differential at each inlet in the time domain in such a way that the dominant inlet flow alternates among them.

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 (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:

 ©2012 COMSOL

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

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

$$p_i = 50 + 10\sin(\pi t + \alpha) \text{ Pa}$$

where t is time (s), and k is a value between zero and one. 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.

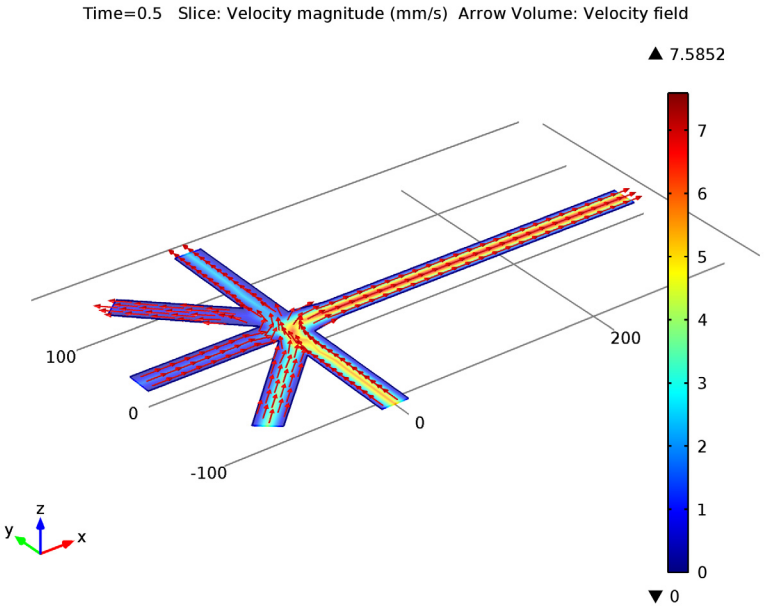

Figure 2: The velocity field in a microchannel infuser through the middle of the geometry.

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

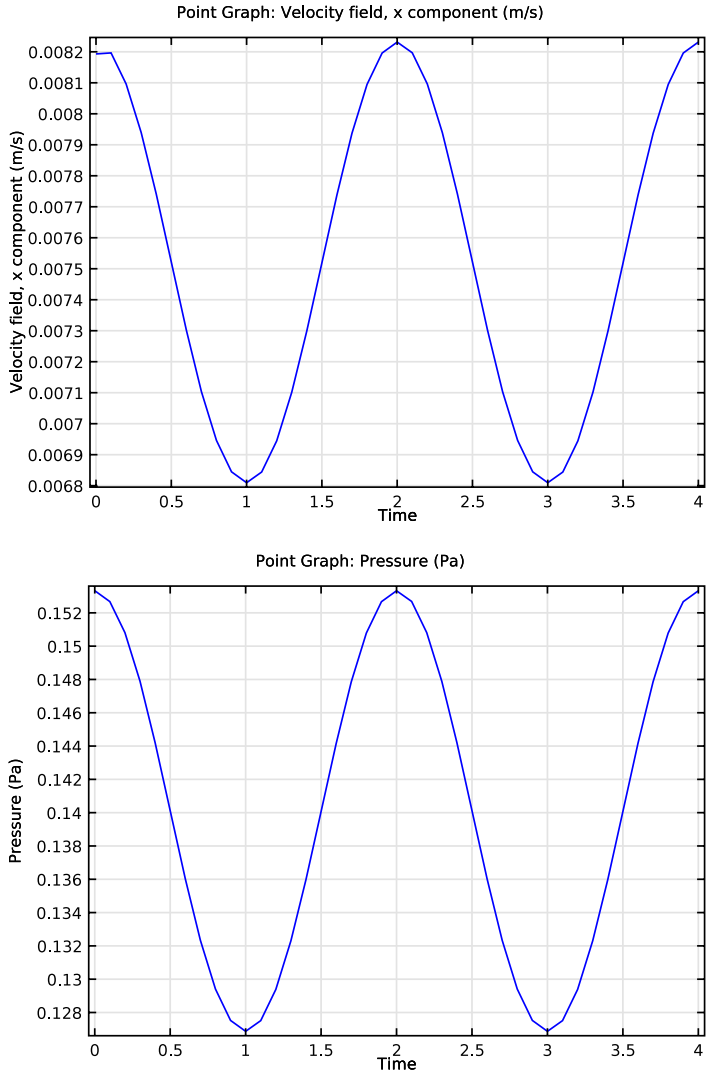

Figure 3: Velocity in the x direction (top) and pressure (bottom) at a point near the outlet.

©2012 COMSOL

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 user 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 tetrahedrons. 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.

Model Library path: `Microfluidics_Module/Fluid_Flow/star_chip`

Modeling Instructions

MODEL WIZARD

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

2 Click **Next**.

3 In the **Add physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

4 Click **Add Selected**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

7 Click **Finish**.

GEOMETRY 1

1 In the **Model Builder** window, under **Model 1** click **Geometry 1**.

2 In the **Geometry** settings window, locate the **Units** section.

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

4 Right-click **Model 1>Geometry 1** and choose **Work Plane**.

Bézier Polygon 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **General** section.

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

4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click the **Add Linear** button.

5 Find the **Control points** subsection. In row **1**, set **xw** to 325.

6 In row **1**, set **yw** to 12.5.

7 In row **2**, set **xw** to 325.

8 In row **2**, set **yw** to 25.

9 Find the **Added segments** subsection. Click the **Add Linear** button.

10 Find the **Control points** subsection. In row **2**, set **xw** to 30.

11 Find the **Added segments** subsection. Click the **Add Linear** button.

12 Find the **Control points** subsection. In row **2**, set **xw** to 0.

13 In row **2**, set **yw** to 30.

14 Find the **Added segments** subsection. Click the **Add Linear** button.

15 Find the **Control points** subsection. In row **2**, set **yw** to 150.

16 Find the **Added segments** subsection. Click the **Add Linear** button.

17 Find the **Control points** subsection. In row **2**, set **xw** to -25.

18 Find the **Added segments** subsection. Click the **Add Linear** button.

19 Find the **Control points** subsection. In row **2**, set **yw** to 55.

20 Find the **Added segments** subsection. Click the **Add Linear** button.

21 Find the **Control points** subsection. In row **2**, set **xw** to -97.

 ©2012 COMSOL

22 In row **2**, set **yw** to 125.

23 Find the **Added segments** subsection. Click the **Add Linear** button.

24 Find the **Control points** subsection. In row **2**, set **xw** to -115.

25 In row **2**, set **yw** to 110.

26 Find the **Added segments** subsection. Click the **Add Linear** button.

27 Find the **Control points** subsection. In row **2**, set **xw** to -30.

28 In row **2**, set **yw** to 25.

29 Find the **Added segments** subsection. Click the **Add Linear** button.

30 Find the **Control points** subsection. In row **2**, set **xw** to -150.

31 Find the **Added segments** subsection. Click the **Add Linear** button.

32 Find the **Control points** subsection. In row **2**, set **yw** to 12.5.

33 Find the **Added segments** subsection. Click the **Add Linear** button.

34 Find the **Control points** subsection. In row **2**, set **xw** to -30.

35 Find the **Added segments** subsection. Click the **Add Linear** button.

36 Locate the **General** section. From the **Type** list, choose **Solid**.

37 Click the **Build Selected** button.

Mirror 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Transforms>Mirror**.

2 Select the object **b1** only.

3 In the **Mirror** settings window, locate the **Input** section.

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

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

6 In the **yw** edit field, type 12.5.

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

8 In the **yw** edit field, type 1.

9 Click the **Build Selected** button.

Union 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Boolean Operations>Union**.

2 Select the objects **b1** and **mir1** only.

3 In the **Union** settings window, locate the **Union** section.

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

5 Click the **Build Selected** button.

Extrude 1

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

2 In the **Extrude** settings window, locate the **Distances from Plane** section.

3 In the table, enter the following settings:

DISTANCES (M)
25

4 Click the **Build All** button.

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

MESH 1

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

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

Size

1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.

2 In the **Size** settings window, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

Distribution 1

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

2 In the **Distribution** settings window, locate the **Distribution** section.

3 In the **Number of elements** edit field, type 4.

4 Select Domain 1 only.

5 Click the **Build All** button.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

 ©2012 COMSOL

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
p0	50[Pa]	Pressure offset
p1	10[Pa]	Pressure amplitude
omega	pi[rad/s]	Angular velocity

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

NAME	EXPRESSION	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

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

2 In the **Material** settings window, locate the **Material Contents** section.

3 In the table, enter the following settings:

PROPERTY	NAME	VALUE
Density	rho	1000
Dynamic viscosity	mu	1e-3

LAMINAR FLOW

Inlet 1

1. In the **Model Builder** window, right-click **Model 1>Laminar Flow** and choose the boundary condition **Inlet**.
2. Select Boundaries 1 and 5 only.
3. In the **Inlet** settings window, locate the **Boundary Condition** section.
4. From the **Boundary condition** list, choose **Pressure, no viscous stress**.
5. Locate the **Pressure, No Viscous Stress** section. In the p_0 edit field, type `p_in_c`.

Inlet 2

1. In the **Model Builder** window, right-click **Laminar Flow** and choose the boundary condition **Inlet**.
2. Select Boundary 7 only.
3. In the **Inlet** settings window, locate the **Boundary Condition** section.
4. From the **Boundary condition** list, choose **Pressure, no viscous stress**.
5. Locate the **Pressure, No Viscous Stress** section. In the p_0 edit field, type `p_in_ir`.

Inlet 3

1. In the **Model Builder** window, right-click **Laminar Flow** and choose the boundary condition **Inlet**.
2. Select Boundary 9 only.
3. In the **Inlet** settings window, locate the **Boundary Condition** section.
4. From the **Boundary condition** list, choose **Pressure, no viscous stress**.
5. Locate the **Pressure, No Viscous Stress** section. In the p_0 edit field, type `p_in_il`.

Inlet 4

1. In the **Model Builder** window, right-click **Laminar Flow** and choose the boundary condition **Inlet**.
2. Select Boundary 14 only.
3. In the **Inlet** settings window, locate the **Boundary Condition** section.
4. From the **Boundary condition** list, choose **Pressure, no viscous stress**.
5. Locate the **Pressure, No Viscous Stress** section. In the p_0 edit field, type `p_in_rm`.

Inlet 5

1. In the **Model Builder** window, right-click **Laminar Flow** and choose the boundary condition **Inlet**.

 ©2012 COMSOL

2 Select Boundary 16 only.

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

4 From the **Boundary condition** list, choose **Pressure, no viscous stress**.

5 Locate the **Pressure, No Viscous Stress** section. In the p_0 edit field, type `p_in_lm`.

Outlet 1

1 In the **Model Builder** window, right-click **Laminar Flow** and choose the boundary condition **Outlet**.

2 Select Boundaries 23 and 24 only.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

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

3 In the **Times** edit field, type `range(0,0.1,4)`.

4 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.

5 Expand the **Study 1>Solver Configurations** node.

Solver 1

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1** node, then click **Time-Dependent Solver 1**.

2 In the **Time-Dependent Solver** settings window, click to expand the **Absolute Tolerance** section.

3 In the **Tolerance** edit field, type `1e-6`.

4 In the **Model Builder** window, right-click **Study 1** and choose **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 **Model Builder** window, under **Results** click **Velocity (spf)**.

2 In the **3D Plot Group** settings window, locate the **Data** section.

3 From the **Time** list, choose **0.5**.

©2012 COMSOL

4 Locate the **Plot Settings** section. Clear the **Plot data set edges** check box.

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

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

6 In the **Slice** settings window, locate the **Expression** section.

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

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

9 In the **Planes** edit field, type 1.

10 Click the **Plot** button.

11 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Arrow Volume**.

12 In the **Arrow Volume** settings window, locate the **Arrow Positioning** section.

13 Find the **x grid points** subsection. In the **Points** edit field, type 35.

14 Find the **y grid points** subsection. In the **Points** edit field, type 35.

15 Find the **z grid points** subsection. In the **Points** edit field, type 1.

16 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Normalized**.

17 Select the **Scale factor** check box.

18 In the associated edit field, type 2600.

19 Click the **Plot** button.

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 **3D Plot Group** settings window, locate the **Data** section.

3 From the **Time** list, choose **0.5**.

4 Click the **Plot** button.

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

Data Sets

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:

1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **Cut Point 3D**.

 ©2012 COMSOL

2 In the **Cut Point 3D** settings window, locate the **Point Data** section.

3 In the **x** edit field, type `324`.

4 In the **y** edit field, type `12.5`.

5 In the **z** edit field, type `12.5`.

1D Plot Group 3

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 In the **1D Plot Group** settings window, locate the **Data** section.

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

4 Right-click **Results>1D Plot Group 3** and choose **Point Graph**.

5 In the **Point Graph** settings window, locate the **Data** section.

6 From the **Data set** list, choose **Cut Point 3D 1**.

7 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Laminar Flow>Velocity field>Velocity field, x component (u)**.

8 Click the **Plot** button.

1D Plot Group 4

1 In the **Model Builder** window, right-click **1D Plot Group 3** and choose **Duplicate**.

2 In the **Model Builder** window, expand the **1D Plot Group 4** node, then click **Point Graph 1**.

3 In the **Point Graph** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Laminar Flow>Pressure (p)**.

4 Click the **Plot** button.

©2012 COMSOL

©2012 COMSOL