Model created in COMSOL Multiphysics 6.4

Micromixer

Introduction

This example studies a laminar static mixer with two parallel sets of split-reshape-recombine mixing elements. The following image shows the geometry of a single mixing element.

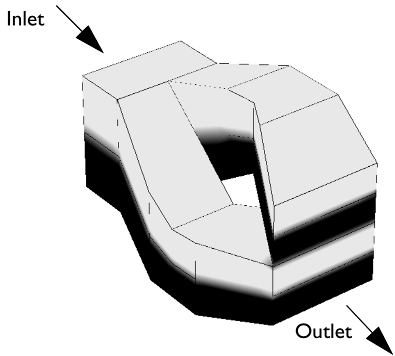


Figure 1: The micromixer splits the incoming fluid in the direction perpendicular to the interface separating the two fluid layers. After recombining them, the mixer stacks the two flows on top of each other, resulting in four fluid layers.

Each mixing element doubles the number of fluid layers, resulting in a fast mixing process. This technique is suited for laminar flow mixing and has small pressure losses. In this example, the mixing structure consists of two parallel sets of mixing elements, where each set is two elements long. You measure the mixing quality with the relative variance of the concentration profile, S, calculated as

$$S_x = \frac{s_x}{s_{\text{inlet}}}$$

$$s_x = \int_{K_x} (c - \bar{c})^2 dA / \int_{K_x} dA$$

where K_x is the yz-plane intersecting the mixing structure at coordinate x, and $\bar{c}$ is the mean concentration.

Model Definition

DOMAIN EQUATIONS

The fluid flow is described by the Navier–Stokes equations

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

where ρ denotes the density (kg/m^3), $\mathbf{u}$ the velocity (m/s), μ denotes the viscosity ($N{\cdot}s/m^2$), and p represents the pressure (Pa). The modeled fluid is water with a viscosity of $1{\cdot}10^{-3}$ $N{\cdot}s/m^2$ and a density of 1000 kg/m^3.

The mass flux is given by diffusion and convection, and the resulting mass balance is

$$\nabla\cdot(-D\nabla c + c\mathbf{u}) = 0$$

where D denotes the diffusion coefficient (m^2/s) and c gives the concentration (mol/m^3). The modeled species has a diffusion coefficient of $5\cdot 10^{-9}$ m^2/s.

BOUNDARY CONDITIONS

At the inlet the model assumes fully developed laminar flow. It sets the velocity to a parabolic profile with a mean velocity of 0.01 m/s. At the outlet, the model sets the pressure to zero. All other boundaries have the no slip condition

$$\mathbf{u} = 0$$

The inlet concentration has a discontinuous profile where the upper half has a concentration of 27 mol/m^3 and the lower half is pure water. The boundary condition is defined such that

$$c\big|_{\text{inlet}} = \begin{cases} c_0 & z > 0 \\ 0 & z \le 0 \end{cases}$$

Results

The figure below shows the final concentration profile. The concentration values in some points of the solution are slightly negative; this is due to the numerical method.

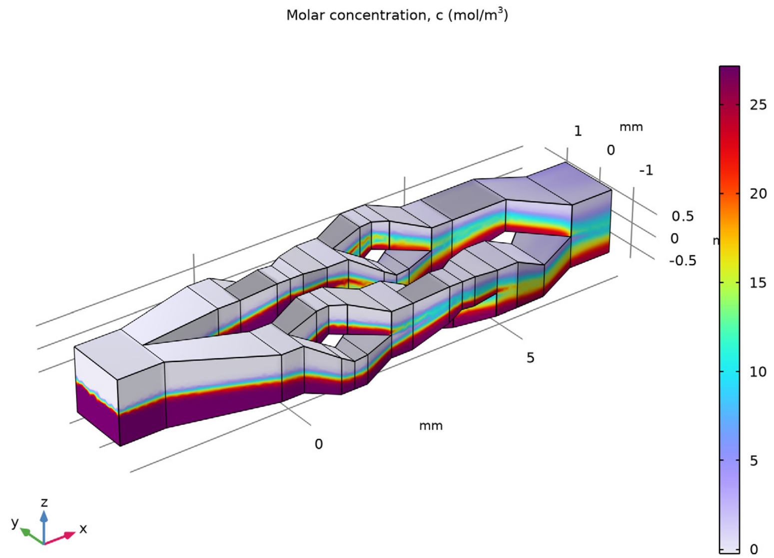


Figure 2: Following three split-reshape-recombine cycles, the outflow has eight fluid layers.

Calculating the relative variance of the concentration at the inlet and outlet, the model set up according to the instructions below reaches a mixing quality of 0.20. For perfect mixing this value would be 0, and a value of 1 means no mixing at all.

Notes About the COMSOL Implementation

The default discretization for the flow equations in the Laminar Flow interface is based on P1+P1 elements. This means that linear elements are used for both velocity and pressure. In this model, the maximum cell Reynolds number for the flow is of the order 10^{-2}, which means that it is beneficial to use a P2+P1 discretization (second order for velocity and first order for pressure).

The default discretization for the concentration in the Transport of Diluted Species interface is linear. This is more robust than a higher-order discretization but requires a refinement of the mesh in order to capture the concentration gradients. This model includes mixing layers with high concentration gradients. In this case it is more computationally efficient to use quadratic elements and a coarser mesh.

Application Library path: `COMSOL_Multiphysics/Fluid_Dynamics/micromixer`

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 In the **Select Physics** tree, select **Chemical Species Transport** > **Transport of Diluted Species (tds)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **General Studies** > **Stationary**.
8 Click **Done**.

GEOMETRY 1

The model geometry is available as a parameterized geometry sequence in a separate MPH file. If you want to create it from scratch yourself, you can follow the instructions in the Appendix — Geometry Modeling Instructions section. Otherwise, insert the geometry sequence as follows:

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Geometry 1** node.
2 Right-click **Geometry 1** and choose **Insert Sequence**.
3 Browse to the model's Application Libraries folder and double-click the file `micromixer_geom_sequence.mph`.

   The application's Application Library folder is shown in the **Application Library path** section immediately before the current section. Note that the path given there is relative to the COMSOL Application Library root, which for a standard installation on Windows is `C:\Program Files\COMSOL\COMSOL64\Multiphysics\Applications`.

4 In the **Geometry** toolbar, click **Build All**.

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

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
`c0`	`27[mol/m^3]`	27 mol/m³	`Inlet concentration`
`D`	`4.5e-9[m^2/s]`	4.5E-9 m²/s	`Diffusion coefficient`
`h_max`	`0.1[mm]`	1E-4 m	`Mesh element size parameter`
`U_mean`	`10[mm/s]`	0.01 m/s	`Mean velocity`
`a`	`1.4[mm]`	0.0014 m	`Inlet width`
`alpha`	`36*U_mean/a^4`	9.3711E10 1/(m³·s)	`Laminar velocity profile normalization constant`

The normalization constant `alpha` applies to a quadratic inlet of the specified side length.

DEFINITIONS

Now define a smoothed step function that you will later use to impose a concentration gradient.

Step 1 (step1)

1 In the **Definitions** toolbar, click f(X) **More Functions** and choose **Step**.

2 In the **Settings** window for **Step**, click to expand the **Smoothing** section.

3 In the **Size of transition zone** text field, type `1e-4`.

Proceed to define global variables for the concentration variances at the outlet and inlet as well as their ratio, `S_outlet`, which gives a measure of the mixing quality. For this purpose, you need two nonlocal average couplings.

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.
2 In the **Settings** window for **Average**, type `aveop_inlet` in the **Operator name** text field.
3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **Inlet**.

Average 2 (aveop2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.
2 In the **Settings** window for **Average**, type `aveop_outlet` in the **Operator name** text field.
3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **Outlet**.

Variables 1

1 In the **Definitions** toolbar, click **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
`Varc_inlet`	`aveop_inlet((c-c0/2)^2)`	mol^2/m^6	Concentration variance, inlet
`Varc_outlet`	`aveop_outlet((c-c0/2)^2)`	mol^2/m^6	Concentration variance, outlet
`S_outlet`	`Varc_outlet/Varc_inlet`		Relative concentration variance, outlet

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)

By default, the first material you add applies to all domains, so you do not need to make any further material settings.

For microfluidic applications, using higher order elements is an efficient way to increase the spatial resolution.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, click to expand the **Discretization** section.

3 From the **Discretization of fluids** list, choose **P2+P1**.

4 Click to collapse the **Discretization** section.

TRANSPORT OF DILUTED SPECIES (TDS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Diluted Species (tds)**.

2 In the **Settings** window for **Transport of Diluted Species**, click to expand the **Discretization** section.

3 From the **Concentration** list, choose **Quadratic**.

4 Click to collapse the **Discretization** section.

LAMINAR FLOW (SPF)

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Laminar Flow (spf)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 Specify the **u** vector as

U_mean	x
0	y
0	z

Inlet 1

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

2 In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Inlet**.

4 Locate the **Velocity** section. Click the **Velocity field** button.

5 Specify the $\mathbf{u}_0$ vector as

`alpha*(a/2+y)*(a/2-y)*(a/2+z)*(a/2-z)`	x
`0`	y
`0`	z

The inlet, of side `a`, is centered around the origin in the *yz*-plane; thus, the expression for `ux` is zero at the channel walls, as required for a fully developed laminar flow profile.

Outlet 1

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

2 In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outlet**.

4 Locate the **Pressure Conditions** section. Select the **Normal flow** checkbox.

TRANSPORT OF DILUTED SPECIES (TDS)

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Transport of Diluted Species (tds)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the c text field, type `c0`.

Inflow 1

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

2 In the **Settings** window for **Inflow**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Inlet**.

4 Locate the **Concentration** section. In the $c_{0,c}$ text field, type `c0*step1(-z[1/m])`.

Outflow 1

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

2 In the **Settings** window for **Outflow**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outlet**.

Fluid 1

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

2 In the **Settings** window for **Fluid**, locate the **Diffusion** section.

3 In the D_c text field, type `D`.

MULTIPHYSICS

Reacting Flow, Diluted Species 1 (rfd1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain** > **Reacting Flow, Diluted Species**.

MESH 1

Modify the physics-induced meshing sequence. The fine mesh close to walls is not needed in this case, but the maximum element size must be restricted to be able to resolve concentration gradients.

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** click **Size**.

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

3 In the **Maximum element size** text field, type `1.7e-1`.

4 In the **Minimum element size** text field, type `7e-2`.

Size 1

In the **Model Builder** window, right-click **Size 1** and choose **Disable**.

Corner Refinement 1

1 In the **Model Builder** window, click **Corner Refinement 1**.

2 In the **Settings** window for **Corner Refinement**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Walls**.

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Walls**.

STUDY 1

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 **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Transport of Diluted Species (tds)**.

Step 2: Stationary 2

1 In the **Study** toolbar, click **Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Laminar Flow (spf)**.

Solution 1 (sol1)

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

2 In the **Settings** window for **Solution**, click **Run**.

RESULTS

Concentration, Streamline (tds)

1 Click the **Zoom Extents** button in the **Graphics** toolbar.

While the first two plot groups visualize the velocity magnitude and the pressure, the third and fourth default plot groups show the concentration as streamline and surface plots. Increase the number of streamlines and add a transparent surface plot with uniform color to highlight the fluid-flow domain.

Streamline 1

1 In the **Model Builder** window, expand the **Concentration, Streamline (tds)** node, then click **Streamline 1**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 In the **Points** text field, type 30.

Surface 1

1 In the **Model Builder** window, right-click **Concentration, Streamline (tds)** and choose **Surface**.

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

3 In the **Expression** text field, type 1.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

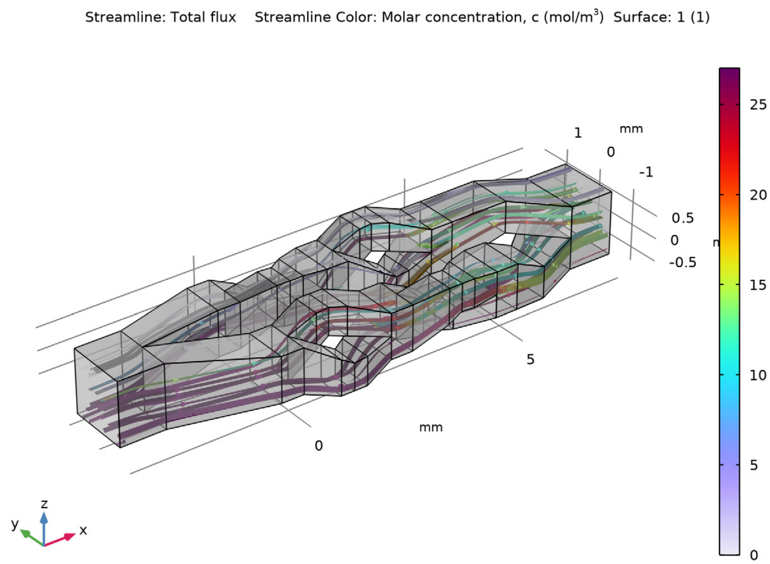
Transparency 1

In the **Model Builder** window, right-click **Surface 1** and choose **Transparency**.

Concentration, Streamline (tds)

1 In the **Concentration, Streamline (tds)** toolbar, click **Plot**.

2 In the **Model Builder** window, under **Results** click **Concentration, Streamline (tds)**.



Concentration, Surface (tds)

1 Click the **Zoom Extents** button in the **Graphics** toolbar.

Compare this surface plot with that in Figure 2.

Global Evaluation 1

Finally, compute the relative concentration variance between the outlet and the inlet to get a measure of the mixing quality.

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Variables** > **S_outlet - Relative concentration variance, outlet - 1**.

3 Click **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

The result should be close to 0.20.

Appendix — Geometry Modeling Instructions

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

NEW

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

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **3D**.

GEOMETRY 1

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

2 From the **Length unit** list, choose **mm**.

Block 1 (blk1)

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

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Depth** text field, type `1.4`.

4 In the **Height** text field, type `1.4`.

5 Locate the **Position** section. In the **x** text field, type `-3.5`.

6 In the **y** text field, type `-0.7`.

7 In the **z** text field, type `-0.7`.

Block 2 (blk2)

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

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Depth** text field, type `1.4`.

4 In the **Height** text field, type `1.4`.

5 Locate the **Position** section. In the **x** text field, type `7`.

6 In the **y** text field, type `-0.7`.

7 In the **z** text field, type `-0.7`.

8 Click **Build Selected**.

9 Click the **Zoom Extents** button in the **Graphics** toolbar.

Block 3 (blk3)

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

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.5`.

4 Locate the **Position** section. In the **x** text field, type `-0.5`.

5 In the **y** text field, type `-1.5`.

6 In the **z** text field, type `-0.5`.

Block 4 (blk4)

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

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.5`.

4 Locate the **Position** section. In the **x** text field, type `2`.

5 In the **y** text field, type `-1.5`.

6 In the **z** text field, type `-0.5`.

Block 5 (blk5)

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

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.5`.

4 Locate the **Position** section. In the **x** text field, type `4.5`.

5 In the **y** text field, type `-1.5`.

6 In the **z** text field, type `-0.5`.

Hexahedron 1 (hex1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Hexahedron**.

2 In the **Settings** window for **Hexahedron**, locate the **Vertices** section:

	x:	y:	z:
1:	-2.5	(As is)	-0.7
2:	-0.5	-1.5	-0.5
3:	-0.5	-0.5	-0.5
4:	-2.5	0	-0.7
5:	-2.5	-0.7	0.7

	x:	y:	z:
6:	-0.5	-1.5	0.5
7:	-0.5	-0.5	0.5
8:	-2.5	0	0.7

Hexahedron 2 (hex2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Hexahedron**.

2 In the table, enter the following settings:

	x:	y:	z:
1:	5	-1.5	-0.5
2:	6	-1.1	-0.8
3:	6	-0.1	-0.8
4:	5	-0.5	-0.5
5:	5	-1.5	0.5
6:	6	-1.1	-0.1
7:	6	-0.1	-0.1
8:	5	-0.5	0.5

Hexahedron 3 (hex3)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Hexahedron**.

2 In the table, enter the following settings:

	x:	y:	z:
1:	6	-1.1	-0.8
2:	7	-0.7	-0.7
3:	7	0.7	-0.7
4:	6	-0.1	-0.8
5:	6	-1.1	-0.1
6:	7	-0.7	0
7:	7	0.7	0
8:	6	-0.1	-0.1

Hexahedron 4 (hex4)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Hexahedron**.

2 In the table, enter the following settings:

	x:	**y:**	**z:**
1:	0	-1.5	-0.5
2:	0.7	-1.7	-0.7
3:	0.7	-1.07	-0.7
4:	0	-1	-0.5
5:	0	-1.5	0.5
6:	0.7	-1.7	0.08
7:	0.7	-1.07	0.08
8:	0	-1	0.5

Hexahedron 5 (hex5)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Hexahedron**.

2 In the table, enter the following settings:

	x:	**y:**	**z:**
1:	0.7	-1.7	-0.7
2:	1	-1.7	-0.7
3:	1	-1.1	-0.7
4:	0.7	-1.07	-0.7
5:	0.7	-1.7	0.08
6:	1	-1.7	-0.1
7:	1	-1.1	-0.1
8:	0.7	-1.07	0.08

Hexahedron 6 (hex6)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Hexahedron**.

2 In the table, enter the following settings:

	x:	**y:**	**z:**
1:	1	-1.7	-0.7
2:	1.3	-1.7	-0.7
3:	1.3	-0.92	-0.7
4:	1	-1.1	-0.7
5:	1	-1.7	-0.1

	x:	y:	z:
6:	1.3	-1.7	-0.07
7:	1.3	-0.92	-0.07
8:	1	-1.1	-0.1

Hexahedron 7 (hex7)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Hexahedron**.

2 In the table, enter the following settings:

	x:	y:	z:
1:	1.3	-1.7	-0.7
2:	2	-1.5	-0.5
3:	2	-0.5	-0.5
4:	1.3	-0.92	-0.7
5:	1.3	-1.7	-0.07
6:	2	-1.5	0
7:	2	-0.5	0
8:	1.3	-0.92	-0.07

Hexahedron 8 (hex8)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Hexahedron**.

2 In the table, enter the following settings:

	x:	y:	z:
1:	2.5	-1.5	-0.5
2:	3.2	-1.7	-0.08
3:	3.2	-1.07	-0.08
4:	2.5	-1	-0.5
5:	2.5	-1.5	0.5
6:	3.2	-1.7	0.7
7:	3.2	-1.07	0.7
8:	2.5	-1	0.5

Hexahedron 9 (hex9)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Hexahedron**.

2 In the table, enter the following settings:

	x:	y:	z:
1:	3.2	-1.7	-0.08
2:	3.5	-1.7	0.1
3:	3.5	-1.1	0.1
4:	3.2	-1.07	-0.08
5:	3.2	-1.7	0.7
6:	3.5	-1.7	0.7
7:	3.5	-1.1	0.7
8:	3.2	-1.07	0.7

Hexahedron 10 (hex10)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Hexahedron**.

2 In the table, enter the following settings:

	x:	y:	z:
1:	3.5	-1.7	0.1
2:	3.8	-1.7	0.07
3:	3.8	-0.92	0.07
4:	3.5	-1.1	0.1
5:	3.5	-1.7	0.7
6:	3.8	-1.7	0.7
7:	3.8	-0.92	0.7
8:	3.5	-1.1	0.7

Hexahedron 11 (hex11)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Hexahedron**.

2 In the table, enter the following settings:

	x:	y:	z:
1:	3.8	-1.7	0.07
2:	4.5	-1.5	0
3:	4.5	-0.5	0
4:	3.8	-0.92	0.07
5:	3.8	-1.7	0.7

	x:	y:	z:
6:	4.5	-1.5	0.5
7:	4.5	-0.5	0.5
8:	3.8	-0.92	0.7

Mirror 1 (mir1)

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

2 Select the objects **hex10**, **hex11**, **hex4**, **hex5**, **hex6**, **hex7**, **hex8**, and **hex9** only.

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

4 Select the **Keep input objects** checkbox.

Mirror 2 (mir2)

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

2 Select the objects **mir1(1)**, **mir1(2)**, **mir1(3)**, **mir1(4)**, **mir1(5)**, **mir1(6)**, **mir1(7)**, and **mir1(8)** only.

3 In the **Settings** window for **Mirror**, locate the **Point on Plane of Reflection** section.

4 In the **y** text field, type -1.

5 Locate the **Normal Vector to Plane of Reflection** section. In the **y** text field, type 1.

6 In the **z** text field, type 0.

Mirror 3 (mir3)

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

2 Select the objects **hex2** and **hex3** only.

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

4 Select the **Keep input objects** checkbox.

Mirror 4 (mir4)

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

2 Select the objects **mir3(1)** and **mir3(2)** only.

3 In the **Settings** window for **Mirror**, locate the **Normal Vector to Plane of Reflection** section.

4 In the **y** text field, type 1.

5 In the **z** text field, type 0.

Mirror 5 (mir5)

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

2 Select the objects **blk4**, **hex10**, **hex11**, and **mir2(2)** only.

3 Select the objects **blk2**, **blk3**, **blk4**, **blk5**, **hex1**, **hex10**, **hex11**, **hex4**, **hex5**, **hex6**, **hex7**, **hex8**, **hex9**, **mir2(1)**, **mir2(2)**, **mir2(3)**, **mir2(4)**, **mir2(5)**, **mir2(6)**, **mir2(7)**, and **mir2(8)** only.

4 In the **Settings** window for **Mirror**, locate the **Normal Vector to Plane of Reflection** section.

5 In the **z** text field, type `0`.

6 In the **y** text field, type `1`.

7 Locate the **Input** section. Select the **Keep input objects** checkbox.

8 Click **Build Selected**.

Union 1 (uni1)

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

2 Click the **Select All** button in the **Graphics** toolbar.

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

4 Clear the **Keep interior boundaries** checkbox.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Inlet

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Inlet` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundary 1 only.

Outlet

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Outlet` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundary 186 only.

Walls

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **x minimum** text field, type `-3.4`.

5 In the **x maximum** text field, type `7.9`.

6 In the **Label** text field, type `Walls`.

7 Click **Build Selected**.