

# Micromixer

# Introduction

This example studies a laminar static mixer with two parallel sets of split-reshaperecombine 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  $\overline{c}$  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  $\rho$  denotes the density (kg/m<sup>3</sup>), **u** the velocity (m/s),  $\mu$  denotes the viscosity (N·s/m<sup>2</sup>), and *p* represents the pressure (Pa). The modeled fluid is water with a viscosity of  $1 \cdot 10^{-3}$  N·s/m<sup>2</sup> and a density of 1000 kg/m<sup>3</sup>.

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  $5 \cdot 10^{-9} \text{ m}^2/s$ .

#### **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<sup>3</sup> 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.



Concentration (mol/m<sup>3</sup>)

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

- I In the Model Wizard window, click 间 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 I

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:

- I In the Model Builder window, expand the Geometry I node.
- 2 Right-click Component I (compl)>Geometry I 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

 $\label{eq:sigma_comsol_comsol_field} Files \verb| COMSOL \verb| COMSOL \verb| COMSOL \verb| Multiphysics \verb| applications. \\$ 

- **4** In the **Geometry** toolbar, click 🛄 **Build All**.
- **5** Click the  $\longleftrightarrow$  **Zoom Extents** button in the **Graphics** toolbar.

# GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.

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	O.1[mm]	IE-4 m	Mesh element size parameter
U_mean	10[mm/s]	0.01 m/s	Mean velocity
а	1.4[mm]	0.0014 m	Inlet width
alpha	36*U_mean/a^4	9.3711E10 1/(m³·s)	Laminar velocity profile normalization constant

**3** In the table, enter the following settings:

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 I (step I)

- I In the Home toolbar, click f(x) Functions and choose Local>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)

- I 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)

- I In the Definitions toolbar, click *P* 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 I

- I In the **Definitions** toolbar, click  $\partial =$  **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	<pre>aveop_inlet((c-c0/2)^2)</pre>	mol²/m <sup>6</sup>	Concentration variance, inlet
Varc_outlet	<pre>aveop_outlet((c-c0/2)^2)</pre>	mol²/m <sup>6</sup>	Concentration variance, outlet
S_outlet	Varc_outlet/Varc_inlet		Relative concentration variance, outlet

#### ADD MATERIAL

- I In the Home 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 Add to Component in the window toolbar.
- 5 In the Home 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)

I In the Model Builder window, under Component I (compl) 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)

- I In the Model Builder window, under Component I (compl) 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

- I In the Model Builder window, under Component I (comp1)>Laminar Flow (spf) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** Specify the **u** vector as

U_mean	x
0	у
0	z

Inlet I

I 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 **u**<sub>0</sub> vector as

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

- I 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 check box.

# TRANSPORT OF DILUTED SPECIES (TDS)

Initial Values 1

- I In the Model Builder window, under Component I (compl)> Transport of Diluted Species (tds) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the c text field, type c0.

## Inflow I

- I 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 I

- I 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**.

## Transport Properties 1

I In the Model Builder window, click Transport Properties I.

2 In the Settings window for Transport Properties, locate the Diffusion section.

**3** In the  $D_{\rm c}$  text field, type D.

# MULTIPHYSICS

Reacting Flow, Diluted Species 1 (rfd1)

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

# MESH I

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.

Size

- I In the Model Builder window, under Component I (comp1) right-click Mesh I and choose Edit Physics-Induced Sequence.
- 2 In the Settings window for Size, click to expand 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 I

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

Corner Refinement I

- I In the Model Builder window, click Corner Refinement I.
- 2 In the Settings window for Corner Refinement, locate the Boundary Selection section.
- 3 From the Selection list, choose Walls.

#### Boundary Layer Properties 1

- I In the Model Builder window, expand the Boundary Layers I node, then click Boundary Layer Properties I.
- **2** In the Settings window for Boundary Layer Properties, locate the Boundary Selection section.
- 3 From the Selection list, choose Walls.

## STUDY I

Step 1: Stationary

I In the Model Builder window, under Study I click Step I: Stationary.

- 2 In the Settings window for Stationary, locate the Physics and Variables Selection section.
- 3 In the table, clear the Solve for check box for Transport of Diluted Species (tds).

#### Stationary 2

- I In the Study toolbar, click **T** Study Steps and choose Stationary>Stationary.
- 2 In the Settings window for Stationary, locate the Physics and Variables Selection section.
- 3 In the table, clear the Solve for check box for Laminar Flow (spf).

Solution 1 (soll)

- I In the Study toolbar, click **Show Default Solver**.
- 2 In the Settings window for Solution, click **=** Compute.

## RESULTS

Concentration, Streamline (tds)

I 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.

Streamline: Total flux Streamline Color: Concentration (mol/m<sup>3</sup>)



Concentration, Surface (tds) 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.

- I In the Results toolbar, click (8.5) 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 I (compl)> Definitions>Variables>S\_outlet Relative concentration variance, outlet.
- 3 Click **=** Evaluate.

# TABLE

I Go to the Table 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 I

- I In the Settings window for Geometry, locate the Units section.
- 2 From the Length unit list, choose mm.

#### Block I (blk1)

- I 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)

- I 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 틤 Build Selected.
- **9** Click the **Com Extents** button in the **Graphics** toolbar.

Block 3 (blk3)

- I 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)

- I 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)

- I 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)

I In the Geometry toolbar, click  $\bigoplus$  More Primitives and choose Hexahedron.

2	In the <b>Settings</b>	window for	or Hexahedron,	locate the	Vertices sect	ion:
---	------------------------	------------	----------------	------------	---------------	------

	x:	y:	<b>z:</b>
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
6:	-0.5	-1.5	0.5
7:	-0.5	-0.5	0.5
8:	-2.5	0	0.7

Hexahedron 2 (hex2)

I In the Geometry toolbar, click  $\bigoplus$  More Primitives and choose Hexahedron.

**2** In the table, enter the following settings:

	x:	у:	<b>Z:</b>
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)

I In the Geometry toolbar, click  $\bigoplus$  More Primitives and choose Hexahedron.

**2** In the table, enter the following settings:

	х:	y:	<b>z:</b>
1:	6	-1.1	-0.8
2:	7	-0.7	-0.7

	<b>x</b> :	y:	z:
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)

I In the Geometry toolbar, click  $\bigoplus$  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)

I In the Geometry toolbar, click  $\bigoplus$  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)

I In the Geometry toolbar, click  $\bigoplus$  More Primitives and choose Hexahedron.

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

**2** In the table, enter the following settings:

Hexahedron 7 (hex7)

I In the Geometry toolbar, click  $\bigoplus$  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)

I In the Geometry toolbar, click  $\bigoplus$  More Primitives and choose Hexahedron.

2	In	the	table,	enter	the	follo	wing	settings:
_			cacie,	encer		10110		oeccingo.

	x:	y:	Z:
1:	2.5	-1.5	-0.5
2:	3.2	-1.7	-0.08
3:	3.2	-1.07	-0.08

	x:	y:	z:
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)

I In the Geometry toolbar, click  $\bigoplus$  More Primitives and choose Hexahedron.

**2** In the table, enter the following settings:

	x:	y:	z:
l:	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)

I In the Geometry toolbar, click  $\bigoplus$  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)

I In the Geometry toolbar, click  $\bigoplus$  More Primitives and choose Hexahedron.

	<b>x</b> :	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
6:	4.5	-1.5	0.5
7:	4.5	-0.5	0.5
8:	3.8	-0.92	0.7

**2** In the table, enter the following settings:

# Mirror I (mirl)

I 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 check box.

#### Mirror 2 (mir2)

- I In the **Geometry** toolbar, click 💭 **Transforms** and choose **Mirror**.
- 2 Select the objects mirl(1), mirl(2), mirl(3), mirl(4), mirl(5), mirl(6), mirl(7), and mirl(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)

- I 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 check box.

#### Mirror 4 (mir4)

- I 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)

- I 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 check box.
- 8 Click 📄 Build Selected.

#### Union I (uni I)

- I 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 check box.

#### Form Union (fin)

- I In the Model Builder window, click Form Union (fin).
- 2 In the Settings window for Form Union/Assembly, click 📳 Build Selected.

#### Inlet

- I 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

- I 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

- I 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 틤 Build Selected.