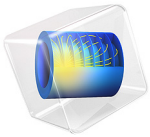

Split and Recombine Mixer Benchmark

Introduction

Benchmarking is an important part of CFD research and code validation. This model uses COMSOL to compute multilamination by a split and recombine mixer. The flow pattern is visualized using a tracer and diffusion is suppressed by setting the diffusion coefficient to an extremely low value of $2 \cdot 10^{-20}$ m^2/s. Thus only numerical diffusion is significant in the model.

This model performs two separate stationary analyses, solving first for the flow and then for the convection/diffusion.

Model Definition

The geometry is constructed with two inlets and a single outlet to allow a flow of high concentration to mix with a flow of low concentration. The two flows primarily mix via multilamination as the flows get split and subsequently recombined as they proceed along the geometry.

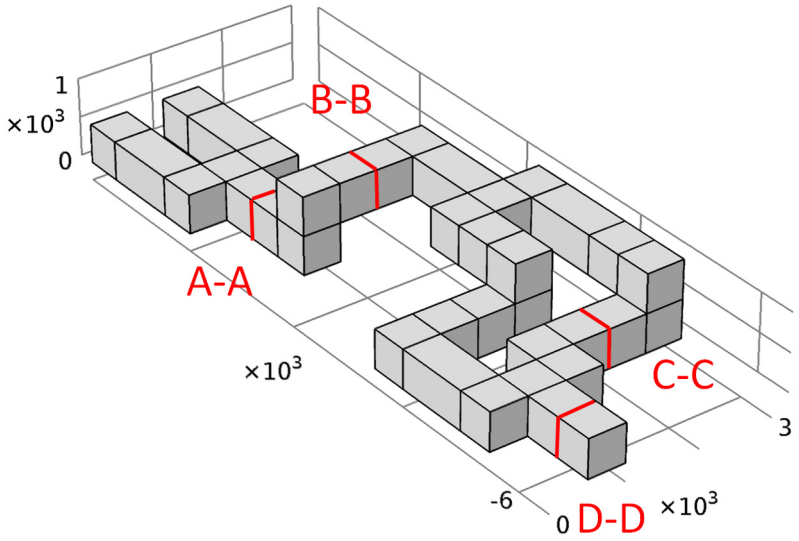

Figure 1: Model geometry. The flow lamination pattern is studied at the cross sections indicated.

The model studies the flow lamination pattern at various places along the mixer channel, labeled A–A, B–B, C–C, and D–D in Figure 1.

Results and Discussion

Figure 2 shows the lamination patterns at the four locations. The blur on the interfaces is due to numerical diffusion within the simulation. High numerical diffusion would result in a more pronounced blurring of the interface. It is clear that the calculated interfaces are sharp indicating little numerical diffusion in the results. These flow patterns are in good agreement with those presented by Glatzel and others (Ref. 1).

Figure 2: Flow lamination patterns at locations A–A, B–B, C–C, and D–D.

The total pressure drop at a given flow rate (fluidic resistance) across the mixer is calculated to be 1.63 Pa. This result is slightly lower that obtained by the other codes in Ref. 1. However a lower value is expected as the finite-element method adds less artificial diffusion to the problem than the finite-volume method. Adding artificial diffusion to a flow problem increases the effective viscosity of the fluid and is therefore expected to artificially increase the pressure drop across the system.

Reference

1. T. Glatzel, C. Litterst, C. Cupelli, T. Lindemann, C. Moosmann, R. Niekrawietz, W. Streule, R. Zengerle, and P. Koltay, "Computational fluid dynamics (CFD) software tools for microfluidic applications - A case study," *Computers & Fluids*, vol. 37, pp. 218–235, 2008.

Notes About the COMSOL Implementation

The model is straightforward to set up using a Laminar Flow interface together with a Transport of Diluted Species interface. The Transport in Diluted Species interface uses the solution from the Laminar Flow interface. This solution coupling is demonstrated in this model.

Application Library path: `Microfluidics_Module/Micromixers/split_recombine_mixer`

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 **Chemical Species Transport>Reacting Flow>Laminar Flow, Diluted Species**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

For convenience, the device geometry is inserted from an existing file. You can read the instructions for creating the geometry in the Appendix — Geometry Instructions.

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `split_recombine_mixer_geom_sequence.mph`.

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

The geometry should look like that in Figure 1.

MATERIALS

Fluid

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

2 In the **Settings** window for **Material**, type `Fluid` in the **Label** text field.

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

Property	Variable	Value	Unit	Property group
Density	rho	977	kg/m³	Basic
Dynamic viscosity	mu	8.55e-4	Pa·s	Basic

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
v0	1[mm/s]	0.001 m/s	Inflow velocity
d	500[um]	5E-4 m	Channel side length

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**.

Two inlets are defined to allow the introduction of the two fluid streams. One stream will carry the tracer.

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. In the U_0 text field, type v0.

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**.

TRANSPORT OF DILUTED SPECIES (TDS)

Streamline and crosswind diffusion are activated by default. There are two options available for the type of crosswind diffusion used to provide numerical stabilization. The default option, **Do Carmo and Galeao** is more effective at suppressing undershoots and overshoots in the concentration, whereas the second option, **Codina** produces less artificial diffusion in the crosswind direction. Make the stabilization settings available following the steps below.

1 Click the **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Stabilization**.

3 Click **OK**.

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

5 In the **Settings** window for **Transport of Diluted Species**, click to expand the **Consistent Stabilization** section.

6 From the **Crosswind diffusion type** list, choose **Codina**.

7 Click to expand the **Discretization** section. From the **Concentration** list, choose **Cubic**.

Choosing cubic discretization for the concentration reduces the numerical diffusion. To suppress physical diffusion, set the diffusion coefficient to an extremely low value. Any diffusion seen in the results will now be due only to numerical diffusion.

Transport Properties 1

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

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

3 In the D_c text field, type `2e-20[m^2/s]`.

Set the initial concentration to 0.5 mol/m^3 to help with numerical convergence when solving.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

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

3 In the c text field, type `0.5`.

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 **Inflow 1**.

Inflow 2

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 **Inflow 2**.

4 Locate the **Concentration** section. In the $c_{0,c}$ text field, type 1.

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**.

MESH 1

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**.

4 Click **Build All**.

STUDY 1

The Transport of Diluted Species interface uses the solution from the Laminar Flow interface. Therefore, only solve for the velocity variables in the first stationary study step and add a second one to compute the concentration of the tracer.

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 table, clear the **Solve for** check box for **Transport of Diluted Species (tds)**.

Stationary 2

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

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

RESULTS

To visualize the lamination at various locations along the mixer, add four **Surface** datasets, labeled A-A, B-B, C-C, and D-D.

A-A

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets** and choose **Surface**.

3 In the **Settings** window for **Surface**, locate the **Parameterization** section.

4 From the **x- and y-axes** list, choose **Expression**.

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

6 Locate the **Selection** section. From the **Selection** list, choose **A-A**.

7 In the **Label** text field, type `A-A`.

B-B

1 Right-click **A-A** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `B-B` in the **Label** text field.

3 Locate the **Parameterization** section. In the **x-axis** text field, type `-y`.

4 Locate the **Selection** section. From the **Selection** list, choose **B-B**.

C-C

1 Right-click **B-B** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `C-C` in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **C-C**.

D-D

1 Right-click **C-C** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `D-D` in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **D-D**.

4 Locate the **Parameterization** section. In the **x-axis** text field, type `x`.

Flow Lamination Pattern

To visualize all lamination results, add four **Surface** plots to the same plot group and then displace them to allow their viewing side by side.

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Flow Lamination Pattern` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Data** section. From the **Dataset** list, choose **A-A**.

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

Contour 1

1 Right-click **Flow Lamination Pattern** and choose **Contour**.

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

3 In the **Expression** text field, type `c`.

4 Locate the **Levels** section. In the **Total levels** text field, type `10`.

5 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

6 Click  **Change Color Table**.

7 In the **Color Table** dialog box, select **Linear>GrayPrint** in the tree.

8 Click **OK**.

Contour 2

1 Right-click **Contour 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **B-B**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Contour 1**.

Translation 1

1 Right-click **Contour 2** and choose **Translation**.

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

3 In the **x** text field, type `d/2`.

4 In the **y** text field, type `-d`.

Contour 3

1 In the **Model Builder** window, under **Results>Flow Lamination Pattern** right-click **Contour 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **C-C**.

Translation 1

1 In the **Model Builder** window, expand the **Contour 3** node, then click **Translation 1**.

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

3 In the **x** text field, type `-5*d`.

4 In the **y** text field, type `0`.

Contour 4

1 In the **Model Builder** window, under **Results>Flow Lamination Pattern** right-click **Contour 3** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **D-D**.

Translation 1

1 In the **Model Builder** window, expand the **Contour 4** node, then click **Translation 1**.

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

3 In the **x** text field, type `4.5*d`.

4 In the **Flow Lamination Pattern** toolbar, click **Plot**.

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

Flow Lamination Pattern

In the **Model Builder** window, under **Results** click **Flow Lamination Pattern**.

Table Annotation 1

1 In the **Flow Lamination Pattern** toolbar, click **More Plots** and choose **Table Annotation**.

2 In the **Settings** window for **Table Annotation**, locate the **Data** section.

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
0.75	-0.075	A-A
1.5	-0.075	B-B
2.25	-0.075	C-C
3	-0.075	D-D

5 Locate the **Coloring and Style** section. Clear the **Show point** check box.

6 From the **Anchor point** list, choose **Center**.

7 In the **Flow Lamination Pattern** toolbar, click **Plot**.

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

9 Click the **Show Legends** button in the **Graphics** toolbar.

10 Click the **Show Grid** button in the **Graphics** toolbar.

Surface Average 1

Calculate the pressure drop across the mixer by adding a **Surface Average** feature to both inlets.

1 In the **Results** toolbar, click **More Derived Values** and choose **Average> Surface Average**.

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

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

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
p	Pa	Pressure

5 Click **Evaluate**.

Appendix — Geometry 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 **Width** text field, type `0.5`.

4 In the **Depth** text field, type `2`.

5 In the **Height** text field, type `0.5`.

Block 2 (blk2)

1 Right-click **Block 1 (blk1)** and choose **Duplicate**.

2 In the **Settings** window for **Block**, locate the **Position** section.

3 In the **x** text field, type `1`.

Block 3 (blk3)

1 Right-click **Block 2 (blk2)** and choose **Duplicate**.

2 In the **Settings** window for **Block**, locate the **Position** section.

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

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

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	1

6 Find the **Layer position** subsection. Clear the **Bottom** check box.

7 Select the **Front** check box.

Block 4 (blk4)

1 Right-click **Block 3 (blk3)** and choose **Duplicate**.

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

3 In the **Width** text field, type `2`.

4 In the **Depth** text field, type `0.5`.

5 Locate the **Position** section. In the **z** text field, type `0.5`.

6 Locate the **Layers** section. Find the **Layer position** subsection. Clear the **Front** check box.

7 Select the **Left** check box.

Block 5 (blk5)

1 Right-click **Block 4 (blk4)** and choose **Duplicate**.

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

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

4 In the **Depth** text field, type `1.5`.

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

6 In the **y** text field, type `-3`.

7 Locate the **Layers** section. Find the **Layer position** subsection. Clear the **Left** check box.

Block 6 (blk6)

1. Right-click **Block 5 (blk5)** and choose **Duplicate**.
2. In the **Settings** window for **Block**, locate the **Position** section.
3. In the **x** text field, type `1.5`.
4. In the **y** text field, type `-4`.

Block 7 (blk7)

1. Right-click **Block 6 (blk6)** and choose **Duplicate**.
2. In the **Settings** window for **Block**, locate the **Size and Shape** section.
3. In the **Depth** text field, type `2.5`.
4. Locate the **Position** section. In the **y** text field, type `-5`.
5. In the **x** text field, type `2.5`.

Block 8 (blk8)

1. Right-click **Block 7 (blk7)** and choose **Duplicate**.
2. In the **Settings** window for **Block**, locate the **Size and Shape** section.
3. In the **Width** text field, type `2`.
4. In the **Depth** text field, type `0.5`.
5. Locate the **Position** section. In the **x** text field, type `0`.
6. In the **y** text field, type `-4`.
7. In the **z** text field, type `0`.

Block 9 (blk9)

1. Right-click **Block 8 (blk8)** and choose **Duplicate**.
2. In the **Settings** window for **Block**, locate the **Position** section.
3. In the **x** text field, type `1`.
4. In the **y** text field, type `-5`.
5. Locate the **Layers** section. Find the **Layer position** subsection. Select the **Left** check box.

Block 10 (blk10)

1. Right-click **Block 9 (blk9)** and choose **Duplicate**.
2. In the **Settings** window for **Block**, locate the **Size and Shape** section.
3. In the **Width** text field, type `1.5`.
4. Locate the **Position** section. In the **x** text field, type `0`.
5. In the **y** text field, type `-5.5`.

6 Locate the **Layers** section. Find the **Layer position** subsection. Clear the **Left** check box.

Block 11 (blk11)

1 Right-click **Block 10 (blk10)** and choose **Duplicate**.

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

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

4 In the **Depth** text field, type `1`.

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

Block 12 (blk12)

1 Right-click **Block 11 (blk11)** and choose **Duplicate**.

2 In the **Settings** window for **Block**, locate the **Position** section.

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

4 In the **y** text field, type `-6.5`.

5 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	`0.5`

6 Find the **Layer position** subsection. Select the **Front** check box.

Partition Domains 1 (pard1)

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

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

3 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.

4 From the **Partition with** list, choose **Extended faces**.

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

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**.

Geometry

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

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

3 From the **Geometric entity level** list, choose **Object**.

4 In the **Label** text field, type `Geometry`.

5 Select the object **fin** only.

Inflow 1

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

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

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

4 In the **Label** text field, type `Inflow 1`.

5 On the object **fin**, select Boundary 101 only.

Inflow 2

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

2 In the **Settings** window for **Explicit Selection**, type `Inflow 2` 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 26 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 28 only.

Inlet

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

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

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Inflow 1** and **Inflow 2**.

6 Click **OK**.

A-A

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

2 In the **Settings** window for **Explicit Selection**, type `A-A` 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 60 only.

B-B

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

2 In the **Settings** window for **Explicit Selection**, type `B-B` 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 125 only.

C-C

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

2 In the **Settings** window for **Explicit Selection**, type `C-C` 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 132 only.

D-D

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

2 In the **Settings** window for **Explicit Selection**, type `D-D` 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 32 only.

Walls - exterior

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

2 In the **Settings** window for **Adjacent Selection**, type `Walls - exterior` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog box, select **Geometry** in the **Input selections** list.

5 Click **OK**.

Walls - interior

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

2 In the **Settings** window for **Adjacent Selection**, type `Walls - interior` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog box, select **Geometry** in the **Input selections** list.

5 Click **OK**.

6 In the **Settings** window for **Adjacent Selection**, locate the **Output Entities** section.

7 Clear the **Exterior boundaries** check box.

8 Select the **Interior boundaries** check box.

Mesh Control

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

2 In the **Settings** window for **Difference Selection**, type `Mesh Control` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, select **Walls - interior** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click **Add**.

9 In the **Add** dialog box, in the **Selections to subtract** list, choose **A-A**, **B-B**, **C-C**, and **D-D**.

10 Click **OK**.

Mesh Control Faces 1 (mcf1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Faces**.

2 In the **Settings** window for **Mesh Control Faces**, locate the **Input** section.

3 From the **Faces to include** list, choose **Mesh Control**.

Exterior Walls

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

2 In the **Settings** window for **Difference Selection**, type `Exterior Walls` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, select **Walls - exterior** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click **Add**.

9 In the **Add** dialog box, in the **Selections to subtract** list, choose **Outlet** and **Inlet**.

10 Click **OK**.