Molecular Flow in an Ion-Implant Vacuum System

Introduction

This example considers the design of an ion implantation system. Ion implantation is used extensively in the semiconductor industry to implant dopants into wafers. Within an ion implanter ions generated within an ion source are accelerated by an electric field to achieve the desired implant energy. Ions of the correct charge state are selected by means of a separation magnet which bends the ion beam to ensure that ions of a particular charge to mass ratio are the only ones which reach the wafer. The energy dose and angle of the ion beam are both key parameters for the process. This part of the system is known as the corrector.

Usually it is desired that only selected regions of the wafer are implanted. This is achieved by masking parts of the wafer with an organic photoresist, to produce the desired pattern. Unfortunately, the photoresist itself emits gas molecules as a result of the beam striking it. These molecules can interact with the ion beam and produce species with undesired charge to mass ratios at different points along the beam path. Some of these species may reach the wafer, degrading the uniformity of the implant, which is highly undesirable. Additionally, these ions may also effect the accuracy of measurements of the implant dose. A key requirement for the system is that the number density of the outgassing molecules for the wafer is low within the beam line.

This example shows how to model an ion implantation system using the Molecular Flow interface. Because it is interactions of outgassing molecules with the beam that produce undesirable species, the average number density of these molecules along the beam path is used as a figure of merit to evaluate the design. Furthermore, because the angle of the wafer to the beam can be modified, the figure of merit must be computed as a function of wafer angle, with rotation about one axis.

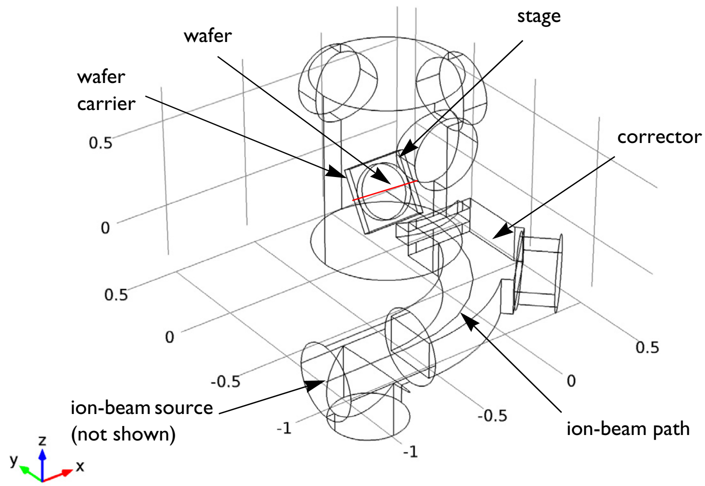

Figure 1: Model geometry. Key components of the system are labeled. The red line through the center of the wafer carrier indicates the axis of rotation of the carrier.

Note: This model is motivated by Ref. 1. However, it is important to note that there are significant differences between the pressure and number density computed by the Molecular Flow interface and those computed in the radiation analogy, which is used in Ref. 1. The COMSOL Multiphysics software computes these quantities accurately, whilst the radiation analogy uses an approximate technique to compute the number density and pressure which can result in misleading, incorrect, answers.

Model Definition

The model geometry is shown in Figure 1. The wafer is positioned on a carrier plate which is rotated about an axis through its center to achieve different implant angles. The carrier plate is mounted in a chamber that is pumped by three large cryopumps, on cylindrical vacuum ports. These pumps have a pump speed of 12,000 l/s. In this model outgassing of only one species from the wafer (H_2) is considered: multiple species can be modeled by adding additional molecular flow interfaces. It is assumed that the outgassing across the wafer surface is uniform and that the total gas emitted is 30 sccm. The vacuum path through the corrector magnetic field enters the main chamber opposite the wafer, and is pumped by a turbomolecular pump on a cylindrical port midway down the corrector, with a pump speed of 1500 l/s and an additional cryopump at the start of the beam path, with a pump speed of 12,000 l/s. There is an aperture at the entrance to the chamber which reduces the flux that enters the corrector. The angle of the wafer normal to the ion beam is swept from 0° to 60° in 20° steps, as the wafer is rotated about the horizontal axis through its center, as shown in Figure 1. All other surfaces in the model are walls.

Results and Discussion

The molecular flux at the surfaces of the vacuum chamber is shown in Figure 2, for wafer normal angles of 0° and 40° to the beam.

The shadowing effect of the wafer support is evident at both wafer angles shown in Figure 2. It is also clear that more flux enters the beam line when the wafer normal is parallel to the beam because the line of sight with the beam line is then parallel to the wafer normal. The number density along the beam path is shown in Figure 3 for each of the angles solved for. The number density increases along the beam path toward the wafer, with strong changes in gradient as the beam passes through different apertures. Note that in the immediate vicinity of the wafer the number density is not computed accurately (this is difficult to see from the plot below, but zooming into the plot at the end of the beam shows issues in the results for elements adjacent to the surface). This is a result of numerical issues when computing the number density very close to a source surface within the domain.

The average number density along the beam path within the corrector can be used as figure of merit for the system design. This quantity is plotted against the angle of the wafer normal to the beam line in Figure 4. The average flux in the corrector is reduced by rotating the wafer normal away from the beam line with a 10% reduction occurring over the full angular range. Rotating the wafer in this manner can therefore result in improved performance of the implanter.

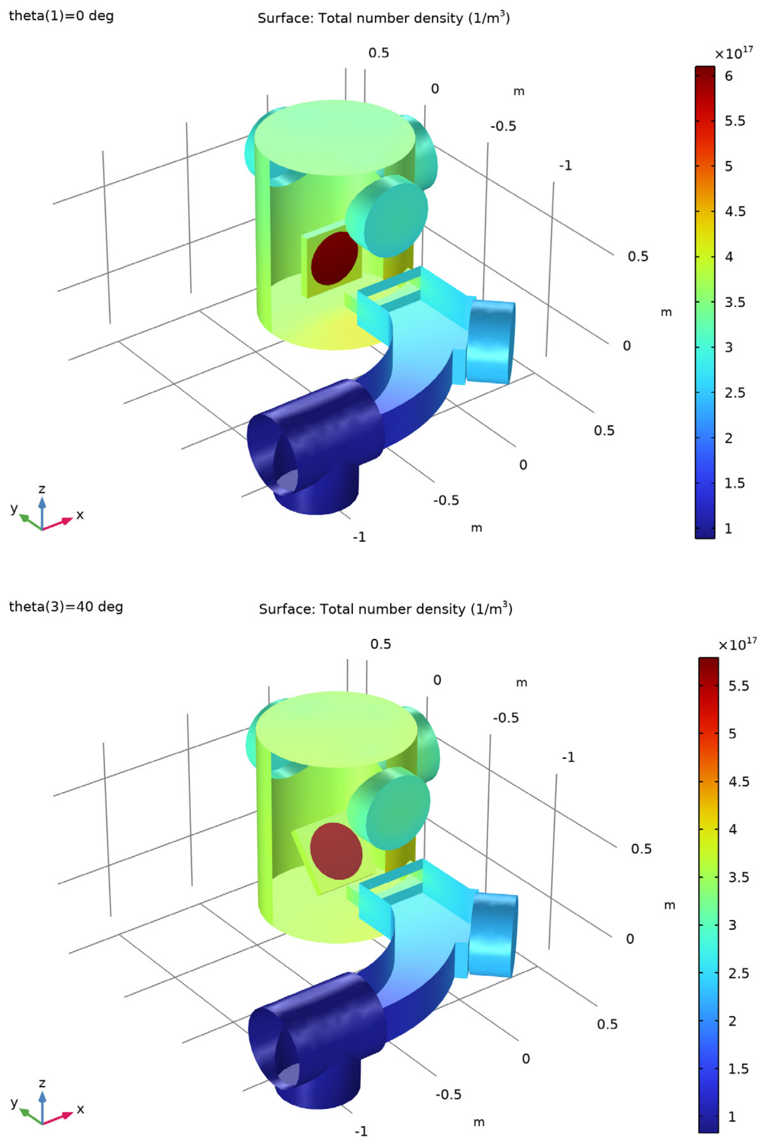

Figure 2: Pressure on the surface of the ion implanter, with the wafer normal at 0° (top) and 40° (bottom) to the beam path.

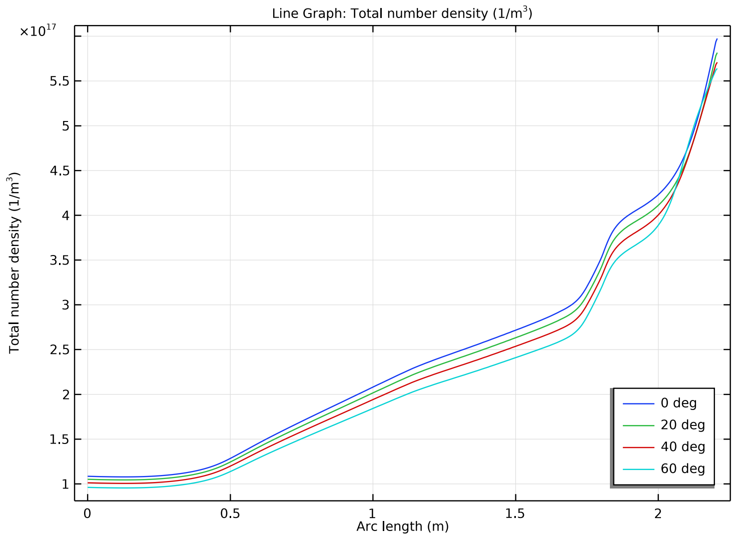

Figure 3: The number density as a function of position along the beam line.

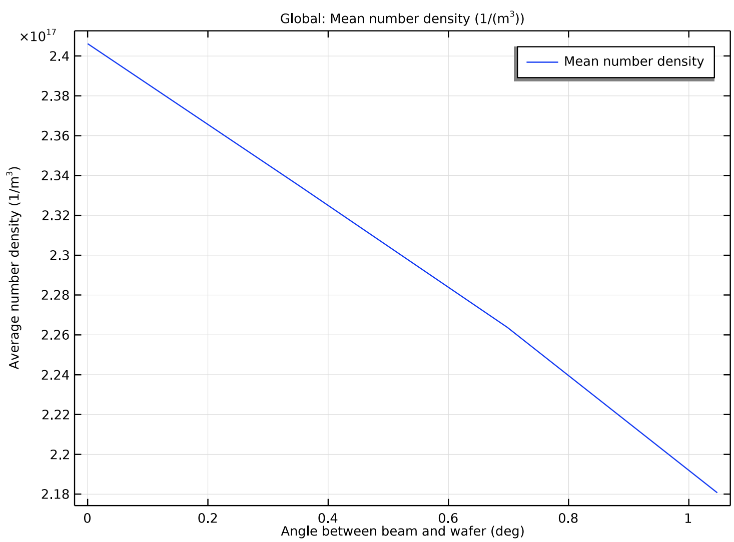

Figure 4: Plot of the average number density along the beam line as a function of the wafer normal angle to the incoming beam.

Reference

1. M.R. LaFontaine, N. Tokoro, P. Murphy, and D. Holbrook, “Modelling Photoresist Outgassing Pressure Distribution Using the Finite Element Method,” *Proc. Conference on Ion Implantation Technology*, IEEE Press, pp. 247–250, 2000.

Notes About the COMSOL Implementation

The model is straightforward to set up using the Molecular Flow interface. The vacuum pump boundary condition is used for the pumps and the outgassing wall boundary condition is used for the wafer surface. The geometry for the vacuum system is imported, but the wafer and its carrier are generated within COMSOL Multiphysics so that the angle to the beam line can be easily parameterized.

Application Library path: `Molecular_Flow_Module/Industrial_Applications/ion_implanter`

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>Rarefied Flow>Free Molecular Flow (fmf)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Insert the prepared geometry sequence from file. You can read the instructions for creating the geometry in the appendix.

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 `ion_implanter_geom_sequence.mph`.

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

GLOBAL DEFINITIONS

Define parameters for the pump speeds. The parameter `theta` for the wafer angle was automatically defined when the geometry sequence was loaded.

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
pumpspeedcryo	12000[l/s]	12 m³/s	Pump speed for cryopumps
pumpspeedturbo	1500[l/s]	1.5 m³/s	Pump speed for turbopump

Add a nonlocal average coupling to compute the average number density along the beam line.

DEFINITIONS

Beam line

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Edges 6, 33, and 103 only.

5 In the **Label** text field, type `Beam line`.

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

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

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

4 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

5 From the **Selection** list, choose **Beam line**.

Set the physics interface to simulate free molecular flow of hydrogen.

FREE MOLECULAR FLOW (FMF)

Molecular Flow 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Free Molecular Flow (fmf)** click **Molecular Flow 1**.

2 In the **Settings** window for **Molecular Flow**, locate the **Molecular Weight of Species** section.

3 In the $M_{\mathrm{n,G}}$ text field, type `0.002[kg/mol]`.

Define the wafer as an outgassing wall that releases 30 sccm of hydrogen.

Wall 2

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

2 Select Boundary 42 only.

3 In the **Settings** window for **Wall**, locate the **Wall Type** section.

4 From the **Wall type** list, choose **Outgassing wall**.

5 Locate the **Flux** section. From the **Outgoing flux** list, choose **Number of SCCM units**.

6 In the $Q_{\mathrm{sccm,G}}$ text field, type `30`.

Set up the vacuum pumps.

Vacuum Pump 1

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

2 In the **Settings** window for **Vacuum Pump**, locate the **Vacuum Pump** section.

3 From the **Specify pump flux** list, choose **Pump speed**.

4 Select Boundary 55 only.

5 In the S_{G} text field, type `pumpspeedturbo`.

Vacuum Pump 2

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

2 In the **Settings** window for **Vacuum Pump**, locate the **Vacuum Pump** section.

3 From the **Specify pump flux** list, choose **Pump speed**.

4 Select Boundary 8 only.

5 In the S_{G} text field, type `pumpspeedcryo`.

Vacuum Pump 3

1 Right-click **Vacuum Pump 2** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundary 25 only.

Vacuum Pump 4

1 Right-click **Vacuum Pump 3** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundary 70 only.

Vacuum Pump 5

1 Right-click **Vacuum Pump 4** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundary 33 only.

Number Density Reconstruction 1

1 In the **Physics** toolbar, click **Edges** and choose **Number Density Reconstruction**.

2 In the **Settings** window for **Number Density Reconstruction**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Beam line**.

Next create the mesh. Add a fine mesh along the beam line.

MESH 1

Edge 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Edge**.

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

3 From the **Selection** list, choose **Beam line**.

Use a very fine mesh along the beam line.

Size 1

1 Right-click **Edge 1** and choose **Size**.

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section.

5 Select the **Maximum element size** check box. In the associated text field, type `0.005`.

Change the global mesh size setting to refine the mesh.

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

Add a free triangular surface mesh. This will inherit the global mesh size setting.

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.
2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All boundaries**.
4 Click **Build All**.

   The domain does not need to be meshed since the interface solves equations only on the surfaces of the model. Note that it is possible to mesh surfaces, edges, and points (as in this model) or to mesh the entire domain. These two approaches cannot be mixed (doing so may lead to incorrect results in some cases).

Set up a parametric sweep over the wafer angle.

STUDY 1

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.
2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
3 Click **Add**.
4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta (Wafer angle)		deg

5 Click **Range**.
6 In the **Range** dialog box, type 0 in the **Start** text field.
7 In the **Step** text field, type 20.
8 In the **Stop** text field, type 60.
9 Click **Add**.
10 In the **Study** toolbar, click **Compute**.

RESULTS

Incident Molecular Flux (fmf)

Duplicate the solution dataset and apply a boundary selection in order to see inside the vacuum chamber.

Study 1/Parametric Solutions 1 (3) (sol2)

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

2 Right-click **Results>Datasets>Study 1/Parametric Solutions 1 (sol2)** and choose **Duplicate**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 From the **Selection** list, choose **Postprocessing**.

Update the default plots to use the new dataset.

Incident Molecular Flux (fmf)

1 In the **Model Builder** window, under **Results** click **Incident Molecular Flux (fmf)**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (3) (sol2)**.

Total Number Density (fmf)

1 In the **Model Builder** window, click **Total Number Density (fmf)**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (3) (sol2)**.

Total Pressure (fmf)

1 In the **Model Builder** window, click **Total Pressure (fmf)**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (3) (sol2)**.

Reproduce the results shown in Figure 2.

Total Number Density (fmf)

1 In the **Model Builder** window, click **Total Number Density (fmf)**.

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

3 From the **Parameter value (theta (deg))** list, choose **0**.

4 In the **Total Number Density (fmf)** toolbar, click **Plot**.

Repeat steps 3 and 4, this time selecting an angle of 40 degrees.

Compare the resulting plot with that in Figure 2.

Create a plot of the number density along the beam path.

Number Density Along Beam Line

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Number Density Along Beam Line` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (2) (sol2)**.

4 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Line Graph 1

1 Right-click **Number Density Along Beam Line** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 From the **Selection** list, choose **Beam line**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Free Molecular Flow>Number density>fmf.ntot - Total number density - 1/m³**.

5 Click to expand the **Legends** section. Select the **Show legends** check box.

6 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

7 In the **Number Density Along Beam Line** toolbar, click **Plot**.

Compare the resulting plot with that in Figure 3.

Finally, plot the mean number density along the beam line, as a function of wafer angle.

Average Number Density vs. Wafer Angle

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Average Number Density vs. Wafer Angle` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (2) (sol2)**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `Angle between beam and wafer (deg)`.

6 Select the **y-axis label** check box. In the associated text field, type Average number density (1/m³).

Global 1

1 Right-click **Average Number Density vs. Wafer Angle** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
aveop1(fmf.ntot)	1/(m^3)	Mean number density

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type theta.

6 In the **Average Number Density vs. Wafer Angle** toolbar, click **Plot**.

Compare the resulting plot with that in Figure 4.

Appendix - Geometry Instructions

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

NEW

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

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
theta	30[deg]	0.5236 rad	Wafer angle

ADD COMPONENT

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

GEOMETRY 1

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type `0.4`.

4 Locate the **Position** section. In the **z** text field, type `-0.3`.

Cylinder 2 (cyl2)

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

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

3 In the **Radius** text field, type `0.2`.

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

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

6 In the **z** text field, type `0.45`.

7 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

Rotate 1 (rot1)

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

2 Select the object **cyl2** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `0,90,180`.

Cylinder 3 (cyl3)

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

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

3 In the **Radius** text field, type `0.2`.

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

5 Locate the **Position** section. In the **x** text field, type `-1.3`.

6 In the **y** text field, type `-1.25`.

7 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

Cylinder 4 (cyl4)

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

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

3 In the **Radius** text field, type `0.2`.

4 In the **Height** text field, type `0.3`.

5 Locate the **Position** section. In the **x** text field, type `-1.05`.

6 In the **y** text field, type `-1.25`.

7 In the **z** text field, type `-0.3`.

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 `0.4`.

4 In the **Depth** text field, type `0.2`.

5 In the **Height** text field, type `0.08`.

6 Locate the **Position** section. In the **x** text field, type `-0.2`.

7 In the **y** text field, type `-0.45`.

8 In the **z** text field, type `-0.04`.

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** check box.

4 From the **Show in physics** list, choose **All levels**.

Work Plane 1 (wp1)>Plane Geometry

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

Work Plane 1 (wp1)>Circle 1 (c1)

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

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

3 In the **Radius** text field, type `0.8`.

4 In the **Sector angle** text field, type `90`.

5 Locate the **Position** section. In the **xw** text field, type `-0.8`.

6 In the **yw** text field, type `-0.45`.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

8 Locate the **Object Type** section. From the **Type** list, choose **Curve**.

Work Plane 1 (wp1)>Rectangle 1 (r1)

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

2 In the **Settings** window for **Rectangle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Width** text field, type `1.3`.

5 In the **Height** text field, type `1.25`.

6 Locate the **Position** section. In the **xw** text field, type `-1.3`.

7 In the **yw** text field, type `-1.25`.

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

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

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

3 In the **Settings** window for **Union**, click **Build Selected**.

Work Plane 1 (wp1)>Delete Entities 1 (del1)

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

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **uni1**, select Boundaries 1 and 3–7 only.

Work Plane 2 (wp2)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the **z-coordinate** text field, type `0.1`.

Work Plane 2 (wp2)>Plane Geometry

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

Work Plane 2 (wp2)>Circle 1 (c1)

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

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `0.95`.

5 In the **Sector angle** text field, type `90`.

6 Locate the **Position** section. In the **xw** text field, type `-0.8`.

7 In the **yw** text field, type `-0.45`.

8 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

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

Layer name	Thickness (m)
`Layer 1`	`0.3`

10 Click **Build Selected**.

Work Plane 2 (wp2)>Delete Entities 1 (del1)

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

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **c1**, select Boundaries 1 and 2 only.

Work Plane 2 (wp2)>Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `0.4`.

4 In the **Height** text field, type `0.36`.

5 Locate the **Position** section. In the **xw** text field, type `-0.2`.

6 In the **yw** text field, type `-0.81`.

Work Plane 2 (wp2)>Rectangle 2 (r2)

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

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

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

4 In the **Height** text field, type `0.25`.

5 Locate the **Position** section. In the **xw** text field, type `-0.12`.

6 In the **yw** text field, type `-1.2`.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `45`.

Work Plane 2 (wp2)>Union 1 (uni1)

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

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

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

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

Extrude 1 (ext1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 2 (wp2)** and choose **Extrude**.

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

3 In the table, enter the following settings:

Distances (m)
0.2

4 Select the **Reverse direction** check box.

Work Plane 3 (wp3)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Face parallel**.

4 On the object **ext1**, select Boundary 10 only.

Work Plane 3 (wp3)>Plane Geometry

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

Work Plane 3 (wp3)>Circle 1 (c1)

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

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

3 In the **Radius** text field, type `0.2`.

Extrude 2 (ext2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 3 (wp3)** and choose **Extrude**.

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

3 In the table, enter the following settings:

Distances (m)
0.2

Union 1 (uni1)

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

2 Select the objects **blk1**, **cyl1**, **cyl3**, **cyl4**, **ext1**, **ext2**, **rot1(1)**, **rot1(2)**, and **rot1(3)** only.

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

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

Work Plane 4 (wp4)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Coordinates**.

4 In row **Point 3**, set **y** to `sin(theta)`.

5 In row **Point 3**, set **z** to `cos(theta)`.

Work Plane 4 (wp4)>Plane Geometry

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

Work Plane 4 (wp4)>Circle 1 (c1)

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

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

3 In the **Radius** text field, type `0.15`.

Work Plane 4 (wp4)>Square 1 (sq1)

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

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

3 In the **Side length** text field, type `0.35`.

4 Locate the **Position** section. From the **Base** list, choose **Center**.

Extrude 3 (ext3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 4 (wp4)** and choose **Extrude**.

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

3 In the table, enter the following settings:

Distances (m)
`0.05`

4 Select the **Reverse direction** check box.

Difference 1 (dif1)

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

2 Select the object **uni1** only.

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

4 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **ext3** only.

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**.

Explicit Selection 1 (sel1)

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 **Boundary**.

4 On the object **fin**, select Boundaries 1, 16, and 18 only.

Postprocessing

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

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

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

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

5 In the **Add** dialog box, select **Explicit Selection 1** in the **Selections to invert** list.

6 Click **OK**.

7 In the **Settings** window for **Complement Selection**, type `Postprocessing` in the **Label** text field.