Model created in COMSOL Multiphysics 6.4

Piezoresistive Pressure Sensor — Layered Shell Version

Introduction

Piezoresistive pressure sensors were some of the first MEMS devices to be commercialized. Compared to capacitive pressure sensors, they are simpler to integrate with electronics, their response is more linear, and they are inherently shielded from RF noise. They do, however, usually require more power during operation, and the fundamental noise limits of the sensor are higher than their capacitive counterparts. Historically, piezoresistive devices have been dominant in the pressure sensor market.

This example considers the design of the MPX100 series pressure sensors originally produced by the semiconductor products division of Motorola Inc. (now Freescale Semiconductor Inc.). Although the sensor is no longer in production, a detailed analysis of its design is given in Ref. 1, and an archived data sheet is available from Freescale Semiconductor Inc. (Ref. 2).

This model is a layered shell version of the *Piezoresistive Pressure Sensor* model in the MEMS Module Application Library and requires the MEMS Module, Structural Mechanics Module, and Composite Materials Module. This tutorial demonstrates how to model piezoresistive effect using the Electric Currents in Layered Shells interface in combination with the Layered Shell interface.

Read more about the Composite Materials Module in the COMSOL blog, Introduction to the Composite Materials Module.

Model Definition

The model consists square membrane with side 1 mm and thickness 20 μm, supported around its edges by region 0.1mm wide, which is intended to represent the remainder of the wafer. The supporting region is fixed on its underside (representing a connection to the thicker handle of the device die). Near to one edge of the membrane an X-shaped piezoresistor (or Xducer™)[1] and part of its associated interconnects are visible. The geometry is shown in Figure 1. The base geometry used to set up the layered physics interfaces is shown in Figure 2.

The piezoresistor is assumed to have a uniform p-type dopant density of 1.32×10^{19} cm^{-3} and a thickness of 400 nm. The interconnects are assumed to have the same thickness but a dopant density of 1.45×10^{20} cm^{-3}. Only a part of the interconnects is included in the

1. Xducer™ is believed to be a trademark of Freescale Semiconductor, Inc. f/k/a Motorola, Inc. Neither Freescale Semiconductor Inc. nor Motorola, Inc. has in any way provided any sponsorship or endorsement of, nor do they have any connection or involvement with, COMSOL Multiphysics® software or this model.

geometry, since their conductivity is sufficiently high that they do not contribute to the voltage output of the device.

The edges of the die are aligned with the {110} directions of the silicon. The die edges are also aligned with the global X- and Y-axes in the COMSOL Multiphysics model. The piezoresistor is oriented at 45° to the die edge, and so lies in the [100] direction of the crystal. In the COMSOL Multiphysics model, a rotated coordinate system with an angle of rotation 45° about the global Z-axis is added to define the orientation of the crystal. Later this rotated coordinate system is used to define the first tangent of a boundary coordinate system which is used in the layered material stack.

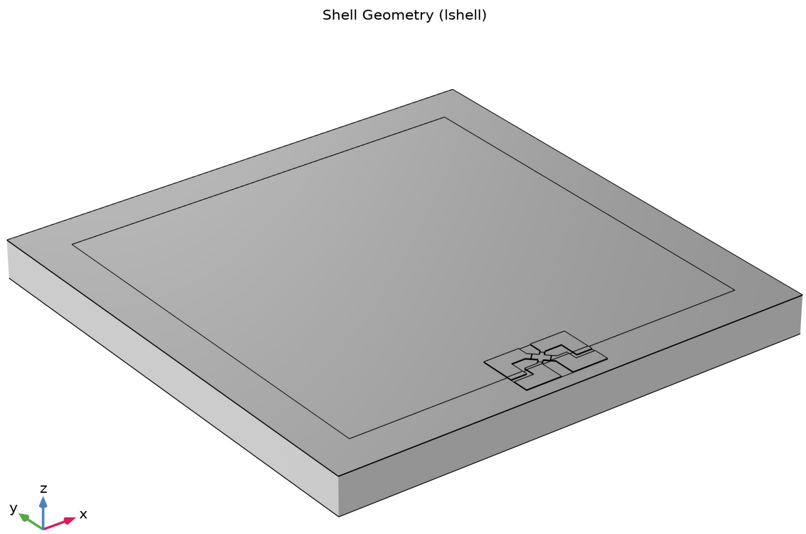

Figure 1: Scaled 3D geometry of piezoresistive pressure sensor.

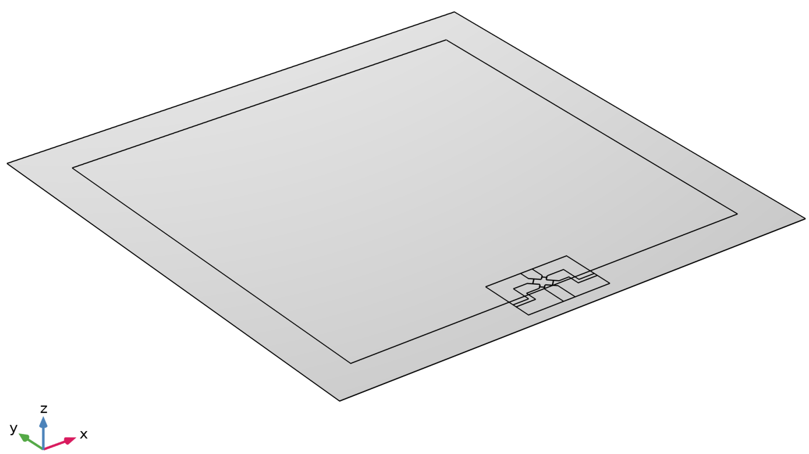

Figure 2: The layered shell version of the model geometry.

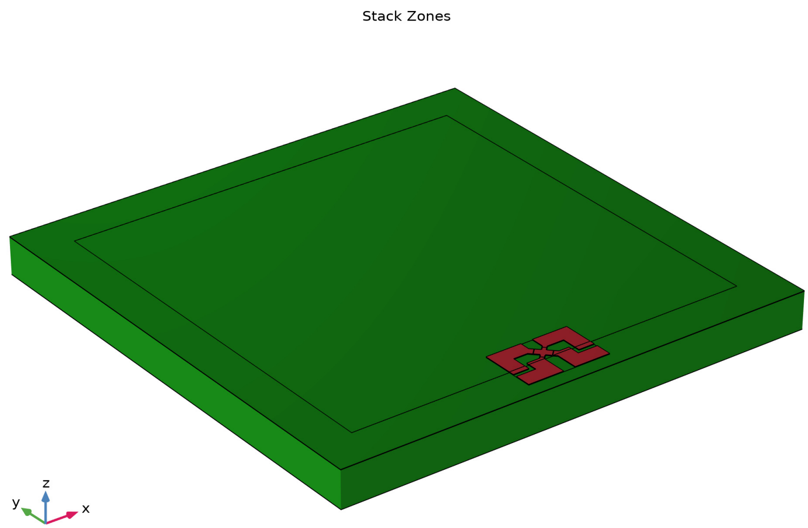

Figure 3: Zones with different layers in the scaled geometry. The green zone has only n-Silicon layer, the red zone has n-Silicon and p-Silicon layers.

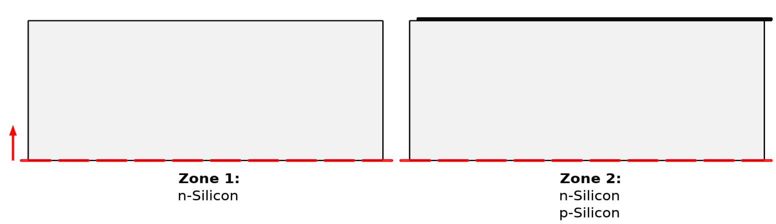

Figure 4: The cross sectional view of zones having layers of different material and thickness.

DEVICE PHYSICS AND EQUATIONS

The conductivity of the Xducer™ sensor changes when the membrane in its vicinity is subject to an applied stress. This effect is known as the piezoresistance effect and is usually associated with semiconducting materials. In semiconductors, piezoresistance results from the strain-induced alteration of the material's band structure, and the associated changes in carrier mobility and number density. The relation between the electric field, $\boldsymbol{E}$, and the current, $\boldsymbol{J}$, within a piezoresistor is:

$$\mathbf{E} = \rho \cdot \mathbf{J} + \Delta\rho \cdot \mathbf{J} \tag{1}$$

where ρ is the resistivity and Δρ is the induced change in the resistivity. In the general case both ρ and Δρ are rank 2 tensors (matrices). The change in resistance is related to the stress, σ, by the constitutive relationship:

$$\Delta\rho = \Pi \cdot \sigma \tag{2}$$

where Π is the piezoresistance tensor (SI units: $\mathrm{Pa}^{-1}\Omega\mathrm{m}$), a material property. Note that the definition of Π in COMSOL Multiphysics includes the resistivity in each element of the tensor, rather than having a scalar multiple outside of Π (which is possible only for materials with isotropic conductivity). Π is in this case a rank-4 tensor; however, it can be represented as a matrix if the resistivity and stress are converted to vectors within a reduced subscript notation. Within the Voigt notation used by COMSOL Multiphysics for this purpose, Equation 2 becomes:

$$\begin{bmatrix} \Delta\rho_{xx} \\ \Delta\rho_{yy} \\ \Delta\rho_{zz} \\ \Delta\rho_{yz} \\ \Delta\rho_{xz} \\ \Delta\rho_{xy} \end{bmatrix} = \begin{bmatrix} \Pi_{11} & \Pi_{12} & \Pi_{13} & \Pi_{14} & \Pi_{15} & \Pi_{16} \\ \Pi_{21} & \Pi_{22} & \Pi_{23} & \Pi_{24} & \Pi_{25} & \Pi_{26} \\ \Pi_{31} & \Pi_{32} & \Pi_{33} & \Pi_{34} & \Pi_{35} & \Pi_{36} \\ \Pi_{41} & \Pi_{42} & \Pi_{43} & \Pi_{44} & \Pi_{45} & \Pi_{46} \\ \Pi_{51} & \Pi_{52} & \Pi_{53} & \Pi_{54} & \Pi_{55} & \Pi_{56} \\ \Pi_{61} & \Pi_{62} & \Pi_{63} & \Pi_{64} & \Pi_{65} & \Pi_{66} \end{bmatrix} \cdot \begin{bmatrix} \sigma^{xx} \\ \sigma^{yy} \\ \sigma^{zz} \\ \sigma^{yz} \\ \sigma^{xz} \\ \sigma^{xy} \end{bmatrix} \tag{3}$$

The Δρ vector computed from Equation 3 is assembled into matrix form in the following manner in Equation 1:

$$\begin{bmatrix} E_x \\ E_y \\ E_z \end{bmatrix} = \begin{bmatrix} \rho_{xx} & \rho_{xy} & \rho_{xz} \\ \rho_{xy} & \rho_{yy} & \rho_{yz} \\ \rho_{xz} & \rho_{yz} & \rho_{zz} \end{bmatrix} \cdot \begin{bmatrix} J_x \\ J_y \\ J_z \end{bmatrix} + \begin{bmatrix} \Delta\rho_{xx} & \Delta\rho_{xy} & \Delta\rho_{xz} \\ \Delta\rho_{xy} & \Delta\rho_{yy} & \Delta\rho_{yz} \\ \Delta\rho_{xz} & \Delta\rho_{yz} & \Delta\rho_{zz} \end{bmatrix} \cdot \begin{bmatrix} J_x \\ J_y \\ J_z \end{bmatrix} \tag{4}$$

Silicon has cubic symmetry, and as a result the Π matrix can be described in terms of three independent constants in the following manner:

$$\Pi = \begin{bmatrix} \Pi_{11} & \Pi_{12} & \Pi_{12} & 0 & 0 & 0 \\ \Pi_{12} & \Pi_{22} & \Pi_{12} & 0 & 0 & 0 \\ \Pi_{12} & \Pi_{12} & \Pi_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & \Pi_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & \Pi_{44} & 0 \\ 0 & 0 & 0 & 0 & 0 & \Pi_{44} \end{bmatrix}$$

For p-type silicon the Π_{44} constant is two orders of magnitude larger than either the Π_{11} or the Π_{12} coefficients. The Π_{66} element (which is equal in magnitude to the Π_{44} element) couples the σ_{xy} shear stress, with the $\Delta\rho_{xy}$ off-diagonal term in the change in resistivity matrix. In turn, $\Delta\rho_{xy}$ couples a current in the *x* direction to an induced electric field in the *y* direction (and vice versa). This is the principle of the Xducer™ transducer. An applied voltage (typically 3 V; see Ref. 2) across the [100] oriented arm of the X produces a current (typically 6 mA; see Ref. 2) down this arm. Shear stresses are present in the Xducer™ as a result of the pressure induced deformation of the diaphragm in which it is implanted. Through the piezoresistance effect, these shear stresses cause an electric field or potential gradient transverse to the direction of current flow, in the [010] arm of the X. Across the width of the transducer, the potential gradient sums up to produce an induced voltage difference between the [010] arms of the X. According to the device data sheet, under normal operating conditions a 60 mV potential difference is generated from a 100 kPa applied pressure with a 3 V applied bias (Ref. 2).

The situation is complicated somewhat by the detailed current distribution within the device, since the voltage sensing elements increase the width of the current carrying silicon wire locally, leading to a “short circuit” effect (Ref. 3) or a spreading out of the current into the sense arms of the X.

Results and Discussion

Figure 5 shows the displacement of the diaphragm as a result of a 100 kPa pressure difference. At the center of the diaphragm the displacement is 1.25 µm. A simple isotropic model for the deform displacement given in Ref. 1 predicts an order of magnitude value of 4 µm (assuming a Young's modulus of 170 GPa and a Poisson's ratio of 0.06). The agreement is reasonable considering the limitations of the analytic model, which is derived by a crude variational guess. A more accurate value for the shear stress in local coordinates at the midpoint of the diaphragm edge is given in Ref. 1 as:

$$\sigma^{l,\,12} = 0{,}141\left(\frac{L}{H}\right)^2 P$$

where P is the applied pressure, L is the length of the diaphragm edge, and H is the diaphragm thickness. This equation predicts the magnitude of the local shear stress to be 35 MPa, in good agreement with the minimum value shown in Figure 6, which is also 34.6 MPa. Theoretically the shear stress should be maximal at the midpoint of the edge of the diaphragm. Figure 7 shows the shear stress along the edge in the model. This shows a maximum magnitude at the center of each of the two edges along which the plot is made.

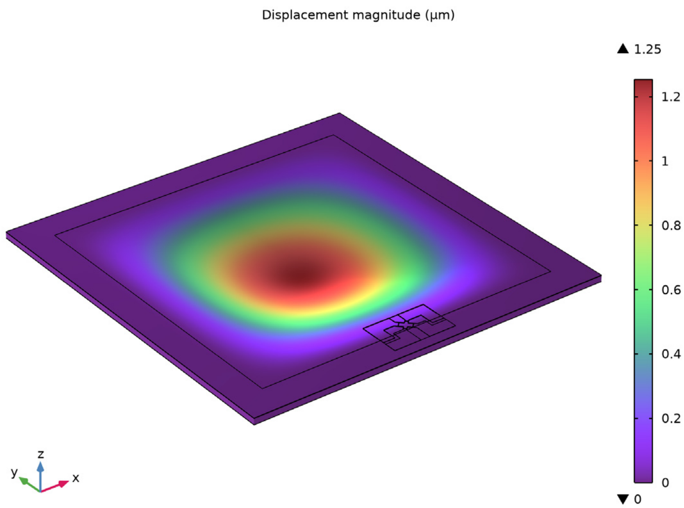

Figure 5: Diaphragm displacement as a result of a 100 kPa applied pressure.

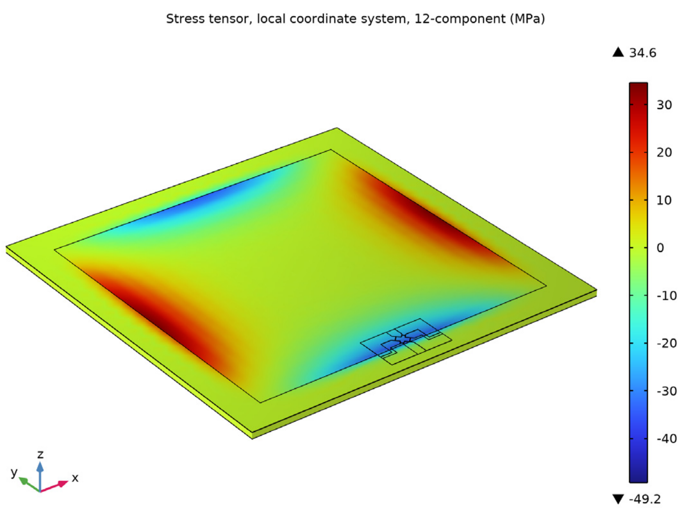

Figure 6: Shear stress, shown in the local coordinate system of the piezoresistor . The shear stress is has its highest magnitude close to the piezoresistor with a value of approximately 34.6 MPa.

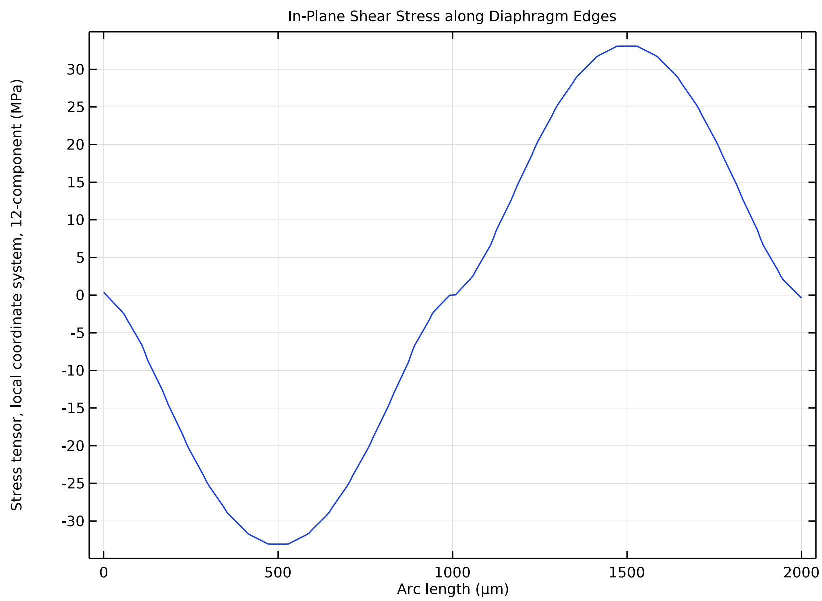

Figure 7: Plot of the local shear stress along two edges of the diaphragm.

The output of the model during normal operation shows good agreement with the manufacturer's data sheet, given that the device dimensions and doping levels have been guessed. With an applied bias of 3 V a typical operating current of 5.9 mA is obtained (compare the current quoted in Ref. 2 of 6 mA). The model produces an output voltage of 52 mV, similar to the actual device output of 60 mV quoted in Ref. 2. The detailed current and voltage distribution within the Xducer™ is shown in Figure 8. There is clear evidence of the current flow "spreading out" into the sense electrodes (which are narrower), a phenomenon described in Ref. 3 as the "short circuit" effect. The asymmetry in the potential, which is induced by the piezoresistive effect, is also apparent in the figure.

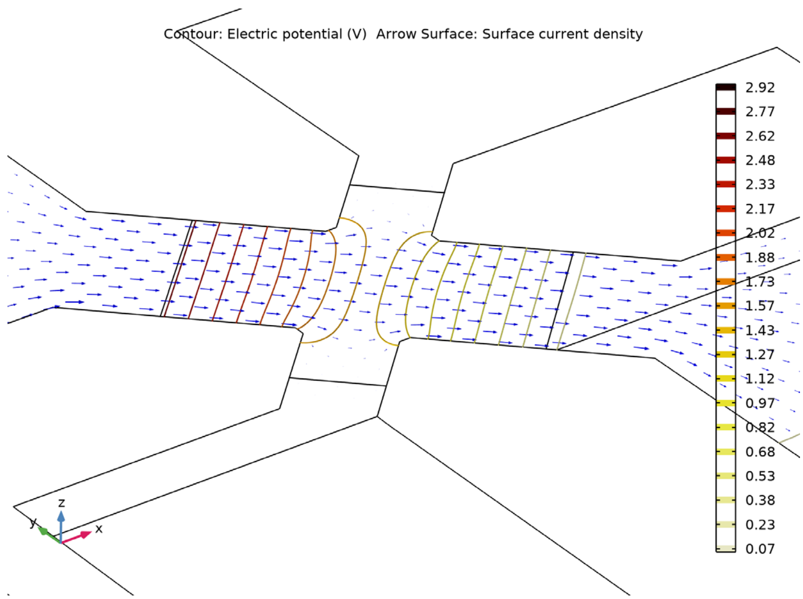

Figure 8: Arrows: Current density, Contours: Electric Potential, for a device driven by a 3 V bias with an applied pressure of 100 kPa.

Notes About the COMSOL Implementation

- Modeling a composite laminated shell requires a 2D surface geometry, called a base surface, and a **Layered Material** node that adds an extra dimension (1D) to the base surface geometry in the surface normal direction. Using the **Layered Material** functionality, you can model several layers of different thicknesses, material properties, and fiber orientations. You can optionally specify the interface materials between the layers and the control mesh elements in each layer.
- The **Layered Material Stack** node is used to define various zones/sections of the piezoresistive pressure sensor.
- The third direction for the selected coordinate system in the **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack** represents the normal direction of the **Layered Shell** or **Shell** physics. This is also the direction in which the layer stacking is interpreted from bottom to top, and therefore, it is crucial to know it during modeling. There are two ways to achieve this:
  - Using physics symbols: Go to the physics settings, find the **Physics Symbols** section, and select the **Enable physics symbols** checkbox. Then go to the material feature, for

instance, **Linear Elastic Material**, to see the normal direction represented by green arrows in the geometry.

  - Using result templates: When a solution dataset is available, use the result template **Thickness and Orientation** to plot the normal direction.

- To visualize the results as a 3D solid object, you can use the **Layered Material** dataset, which creates a virtual 3D solid object combining the surface geometry (2D) and the extra dimension (1D).

References

1. S.D. Senturia, “A Piezoresistive Pressure Sensor,” *Microsystem Design*, chapter 18, Springer, 2000.

2. Motorola Semiconductor MPX100 series technical data, document: MPX100/D, 1998 (available from Freescale Semiconductor Inc at www.nxp.com).

3. M. Bao, *Analysis and Design Principles of MEMS Devices*, Elsevier B.V., 2005.

Application Library path: `Composite_Materials_Module/Multiphysics/piezoresistive_pressure_sensor_layered`

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 **Structural Mechanics** > **Electromagnetics–Structure Interaction** > **Piezoresistivity** > **Piezoresistivity, Layered Shell**.
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.

The dimensions of this geometry are given in micrometers, so you need to change the length unit accordingly.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

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

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

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

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

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

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

Some selections in the inserted geometry need to be modified for the current model setup.

Model Boundaries (unisel1)

In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Model Boundaries (unisel1)** and choose **Delete**.

Model Boundaries

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

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

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

4 Select the **Group by continuous tangent** checkbox.

5 On the object **ige2**, select Boundaries 1–18 only.

6 Drag and drop below **Membrane (boxsel1)**.

7 Click **Build Selected**.

Fixed Edges (adjsel1)

In the **Model Builder** window, right-click **Fixed Edges (adjsel1)** and choose **Delete**.

Fixed Boundaries (difsel1)

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

2 Click **Build Preceding State**.

3 Click the **Add** button for **Selections to add**.

4 In the **Add** dialog, select **Model Boundaries** in the **Selections to add** list.

5 Click **OK**.

6 In the **Settings** window for **Difference Selection**, click **Build Selected**.

DEFINITIONS

Rotated System 2 (sys2)

1 In the **Definitions** toolbar, click **Coordinate Systems** and choose **Rotated System**.

2 In the **Settings** window for **Rotated System**, locate the **Rotation** section.

3 Find the **Euler angles** subsection. In the α text field, type `-45[deg]`.

Setup the first axis of the boundary coordinate system from the rotated system.

Boundary System 1 (sys1)

1 In the **Model Builder** window, click **Boundary System 1 (sys1)**.

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

3 Find the **Coordinate names** subsection. From the **Create first tangent direction from** list, choose **Rotated System 2 (sys2)**.

4 From the **Axis** list, choose **x1**.

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 **Piezoresistivity** > **n-Silicon (single-crystal, lightly doped)**.

4 Click the **Add to Global Materials** button in the window toolbar.

5 In the tree, select **Piezoresistivity** > **p-Silicon (single-crystal, lightly doped)**.

6 Click the **Add to Global Materials** button in the window toolbar.

7 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Layered Material Stack 1 (stlmat1)

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

Layered Material Link 1 (stlmat1.stllmat1)

In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** > **Layered Material Stack 1 (stlmat1)** right-click **Layered Material Link 1 (stlmat1.stllmat1)** and choose **Delete**.

Layered Material Stack 1 (stlmat1)

1 In the **Settings** window for **Layered Material Stack**, locate the **Orientation and Position** section.

2 From the **Position** list, choose **Bottom side on boundary**.

n-Silicon

1 Right-click **Layered Material Stack 1 (stlmat1)** and choose **Layered Material**.

2 In the **Settings** window for **Layered Material**, type `n-Silicon` in the **Label** text field.

3 Locate the **Layer Definition** section. In the table, enter the following settings:

Layer	Material	Rotation	Value (m)	Thickness	Mesh elements
`Layer 1`	n-Silicon (single-crystal, lightly doped) (mat1)	`0.0`	0 rad	`20[um]`	1

p-Silicon

1 Right-click **Layered Material Stack 1 (stlmat1)** and choose **Layered Material**.

2 In the **Settings** window for **Layered Material**, type `p-Silicon` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Electric currents**.

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

Layer	Material	Rotation	Value (m)	Thickness	Mesh elements
`Layer 1`	p-Silicon (single-crystal, lightly doped) (mat2)	`0.0`	0 rad	`400[nm]`	1

Layered Material Stack 1 (stlmat1)

1 In the **Model Builder** window, click **Layered Material Stack 1 (stlmat1)**.

2 In the **Settings** window for **Layered Material Stack**, click **Layer Cross-Section Preview** in the upper-right corner of the **Layered Material Settings** section. From the menu, choose **Create Layer Cross-Section Plot**.

LAYERED SHELL (LSHELL)

Linear Elastic Material 1

1 In the **Model Builder** window, expand the **Results** node, then click **Component 1 (comp1)** > **Layered Shell (lshell)** > **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.

3 From the **Material symmetry** list, choose **Anisotropic**.

4 From the **Material data ordering** list, choose **Voigt (11, 22, 33, 23, 13, 12)**.

Fixed Constraint, Interface 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint, Interface**.

2 In the **Settings** window for **Fixed Constraint, Interface**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Fixed Boundaries**.

4 Locate the **Interface Selection** section. From the **Apply to** list, choose **Bottom interface**.

Face Load 1

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

2 In the **Settings** window for **Face Load**, locate the **Interface Selection** section.

3 From the **Apply to** list, choose **Top interface**.

4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.

5 In the p text field, type `100[kPa]`.

6 Locate the **Boundary Selection** section. From the **Selection** list, choose **Membrane**.

Continuity 1

1 In the **Physics** toolbar, click **Edges** and choose **Continuity**.

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

3 From the **Source** list, choose **Layered Material Stack 1 (stlmat1.zone1)**.

4 From the **Destination** list, choose **Layered Material Stack 1 (stlmat1.zone2)**.

ELECTRIC CURRENTS IN LAYERED SHELLS (ECIS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents in Layered Shells (ecis)**.

2 In the **Settings** window for **Electric Currents in Layered Shells**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Electric currents**.

4 Locate the **Shell Properties** section. Clear the **Use all layers** checkbox.

5 In the **Selection** table, clear the checkbox for **Layer 1 - n-Silicon**.

The electric conductivity is a function of number density. To change default value of number density, activate the **Model Input** sections in the **Conductive Shell 1** and **Piezoresistive Shell 1** features.

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

7 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

8 Click **OK** to show model input sections.

Conductive Shell 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Electric Currents in Layered Shells (ecis)** click **Conductive Shell 1**.

2 In the **Settings** window for **Conductive Shell**, click to expand the **Model Input** section.

3 In the n_d text field, type `1.45e20[1/cm^3]`.

Piezoresistive Shell 1

1 In the **Model Builder** window, click **Piezoresistive Shell 1**.

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

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

4 Click to expand the **Model Input** section. In the n_d text field, type `1.32e19[1/cm^3]`.

Conductive Shell 1

In the **Model Builder** window, click **Conductive Shell 1**.

Terminal 1

1 In the **Physics** toolbar, click **Attributes** and choose **Terminal**.

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

3 Click **Clear Selection**.

4 Select Edges 11 and 13 only.

5 Locate the **Terminal** section. From the **Terminal type** list, choose **Voltage**.

6 In the V_0 text field, type 3.

Conductive Shell 1

In the **Model Builder** window, click **Conductive Shell 1**.

Terminal 2

1 In the **Physics** toolbar, click **Attributes** and choose **Terminal**.

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

3 Click **Clear Selection**.

4 Select Edge 7 only.

Conductive Shell 1

In the **Model Builder** window, click **Conductive Shell 1**.

Terminal 3

1 In the **Physics** toolbar, click **Attributes** and choose **Terminal**.

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

3 Click **Clear Selection**.

4 Select Edges 72 and 73 only.

Ground 1

1 In the **Physics** toolbar, click **Edges** and choose **Ground**.

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

3 Click **Clear Selection**.

4 Select Edge 70 only.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled 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 Parameters** section.

3 In the **Maximum element size** text field, type 60.

4 In the **Minimum element size** text field, type 0.5.

Size 1

1 Right-click **Size** and choose **Duplicate**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **Piezoresistor**.
5 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `2`.
6 In the **Minimum element size** text field, type `0.1`.

Size 2

1 Right-click **Size 1** and choose **Duplicate**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Selection** list, choose **Connections**.
4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `6`.

Size 3

1 Right-click **Size 2** and choose **Duplicate**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Edge**.
4 Select Edges 27, 29, 38–44, and 51–53 only.
5 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.4`.

Free Triangular 1

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Move Down** three times.
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**.

STUDY 1

Switch off the generation of default plots in the study. We will use **Result Templates**.

1 In the **Model Builder** window, click **Study 1**.
2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** checkbox.

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

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	MPa

8 Click **Apply**.

Now create a dataset for the detailed current and voltage distribution plots.

Study 1/Solution 1 (2) (sol1)

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

2 Right-click **Results** > **Datasets** > **Study 1/Solution 1 (sol1)** 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 **Electric currents**.

Layered Material 1

1 In the **Results** toolbar, click **More Datasets** and choose **Layered Material**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.

Use the result templates to show the von Mises stress, displacement and voltage distribution.

RESULT TEMPLATES

1 In the **Home** toolbar, click **Windows** and choose **Result Templates**.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (1) (sol1)** > **Layered Shell** > **von Mises Stress (lshell)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the tree, select **Study 1/Solution 1 (1) (sol1)** > **Layered Shell** > **Displacement (lshell)**.

6 Click the **Add Result Template** button in the window toolbar.

7 In the tree, select **Study 1/Solution 1 (1) (sol1)** > **Electric Currents in Layered Shells** > **Electric Potential (ecis)**.

8 Click the **Add Result Template** button in the window toolbar.

9 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

von Mises Stress (lshell)

1 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

2 Select the **Show maximum and minimum values** checkbox.

Surface 1

1 In the **Model Builder** window, expand the **von Mises Stress (lshell)** node, then click **Surface 1**.

2 In the **von Mises Stress (lshell)** toolbar, click **Plot**.

Displacement (lshell)

1 In the **Model Builder** window, under **Results** click **Displacement (lshell)**.

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

3 Select the **Show maximum and minimum values** checkbox.

4 In the **Displacement (lshell)** toolbar, click **Plot**.

Electric Potential (ecis)

1 In the **Model Builder** window, click **Electric Potential (ecis)**.

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

3 From the **Dataset** list, choose **Layered Material 1**.

4 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

5 In the **Electric Potential (ecis)** toolbar, click **Plot**.

Now create a plot of the shear stress in the local coordinate system of the piezoresistor.

In-Plane Shear Stress

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `In-Plane Shear Stress` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material**.

4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

Surface 1

1 Right-click **In-Plane Shear Stress** and choose **Surface**.

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

3 In the **Expression** text field, type `lshell.slGp12`.

4 In the **In-Plane Shear Stress** toolbar, click **Plot**.

Create a line plot of the shear stress in the local coordinate system of the piezoresistor.

In-Plane Shear Stress along Diaphragm Edges

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

2 In the **Settings** window for **1D Plot Group**, type `In-Plane Shear Stress along Diaphragm Edges` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** checkbox. In the associated text field, type `Stress tensor, local coordinate system, 12-component (MPa)`.

Line Graph 1

1 Right-click **In-Plane Shear Stress along Diaphragm Edges** and choose **Line Graph**.

2 Select Edges 6 and 76 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `side(2,lshell.atxd1(lshell.d, lshell.slGp12))`.

5 From the **Unit** list, choose **MPa**.

6 In the **In-Plane Shear Stress along Diaphragm Edges** toolbar, click **Plot**.

Now create a plot of the detailed current and voltage distribution.

Current and Voltage

1. In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2. In the **Settings** window for **3D Plot Group**, type `Current and Voltage` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.
4. Locate the **Plot Settings** section. From the **View** list, choose **New view**.

Contour 1

1. Right-click **Current and Voltage** and choose **Contour**.
2. In the **Settings** window for **Contour**, locate the **Expression** section.
3. In the **Expression** text field, type `V`.
4. Locate the **Levels** section. In the **Total levels** text field, type `20`.
5. Locate the **Coloring and Style** section. From the **Contour type** list, choose **Tube**.
6. Select the **Radius scale factor** checkbox.
7. In the **Tube radius expression** text field, type `1`.
8. In the **Radius scale factor** text field, type `0.07`.
9. From the **Color table** list, choose **ThermalDark**.
10. From the **Color table transformation** list, choose **Reverse**.

Arrow Surface 1

1. In the **Model Builder** window, right-click **Current and Voltage** and choose **Arrow Surface**.
2. In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Electric Currents in Layered Shells** > **Currents and charge** > **ecis.JsX,...,ecis.JsZ - Surface current density (material and geometry frames)**.
3. Locate the **Expression** section.
4. Select the **Description** checkbox. In the associated text field, type `Surface current density`.
5. Locate the **Coloring and Style** section.
6. Select the **Scale factor** checkbox. In the associated text field, type `0.005`.
7. Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `3000`.
8. Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.
9. In the **Current and Voltage** toolbar, click **Plot**.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (1) (sol1)** > **Layered Shell** > **Geometry and Layup (lshell)** > **Shell Geometry (lshell)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Shell Geometry (lshell)

Click the **Go to Default View** button in the **Graphics** toolbar.

Layered Material 2 (Shell Geometry)

1 In the **Model Builder** window, under **Results** > **Datasets** click **Layered Material 2 (Shell Geometry)**.

2 In the **Settings** window for **Layered Material**, locate the **Layers** section.

3 In the **Scale** text field, type `5`.

Shell Geometry (lshell)

1 In the **Model Builder** window, under **Results** click **Shell Geometry (lshell)**.

2 In the **Shell Geometry (lshell)** toolbar, click **Plot**.

Stack Zones

1 Right-click **Shell Geometry (lshell)** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Stack Zones` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material 2 (Shell Geometry)**.

Surface 1

1 In the **Model Builder** window, expand the **Stack Zones** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `stlmat1.zone`.

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

5 From the **Color table** list, choose **TrafficLight**.

Stack Zones

1 In the **Model Builder** window, click **Stack Zones**.

2 In the **Stack Zones** toolbar, click **Plot**.

3 Drag and drop below **Layer Cross-Section Preview**.

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 **µm**.

Work Plane 1 (wp1)

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

Work Plane 1 (wp1) > Plane Geometry

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

Work Plane 1 (wp1) > 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 1200.

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

5 In the **yw** text field, type 478.

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

Layer name	Thickness (µm)
Layer 1	100

7 Select the **Layers to the left** checkbox.

8 Select the **Layers to the right** checkbox.

9 Select the **Layers on top** checkbox.

Work Plane 1 (wp1) > Square 2 (sq2)

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 `22.6`.

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

Work Plane 1 (wp1) > 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 `52.5`.

4 In the **Height** text field, type `10`.

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

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

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

Layer name	Thickness (µm)
Layer 1	6.25

8 Select the **Layers to the right** checkbox.

9 Select the **Layers to the left** checkbox.

10 Clear the **Layers on bottom** checkbox.

Work Plane 1 (wp1) > Rectangle 2 (r2)

1 Right-click **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** > **Plane Geometry** > **Rectangle 1 (r1)** and choose **Duplicate**.

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

3 In the **Width** text field, type `62.5`.

4 In the **Height** text field, type `20`.

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

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

Layer name	Thickness (µm)
Layer 1	11.25

Work Plane 1 (wp1) > Rectangle 3 (r3)

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 210.

4 In the **Height** text field, type 140.

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

6 In the **yw** text field, type -15.

Work Plane 1 (wp1) > Rectangle 4 (r4)

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 40.

4 In the **Height** text field, type 50.

5 Locate the **Position** section. In the **xw** text field, type -105.

6 In the **yw** text field, type -35.

Work Plane 1 (wp1) > Rectangle 5 (r5)

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 90.

4 In the **Height** text field, type 40.

5 Locate the **Position** section. In the **xw** text field, type -105.

6 In the **yw** text field, type 15.

Work Plane 1 (wp1) > Rectangle 6 (r6)

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 90.

4 In the **Height** text field, type 40.

5 Locate the **Position** section. In the **xw** text field, type -105.

6 In the **yw** text field, type -85.

Work Plane 1 (wp1) > Rectangle 7 (r7)

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 40.

4 In the **Height** text field, type 30.

5 Locate the **Position** section. In the **xw** text field, type `-55`.

6 In the **yw** text field, type `-45`.

Work Plane 1 (wp1) > Mirror 1 (mir1)

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

2 Select the objects **r4**, **r5**, **r6**, and **r7** only.

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

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

Form Composite Faces 1 (cmf1)

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

2 On the object **fin**, select Boundaries 19, 20, 26–35, and 38 only.

Form Composite Faces 2 (cmf2)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

2 On the object **cmf1**, select Boundaries 11, 14–18, 25, 27, and 32–35 only.

Ignore Edges 1 (ige1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **cmf2**, select Edges 24 and 94 only.

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

4 Clear the **Ignore adjacent vertices** checkbox.

Form Composite Faces 3 (cmf3)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

2 On the object **ige1**, select Boundaries 7, 12, 15, 17, 21, and 24–26 only.

Ignore Edges 2 (ige2)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **cmf3**, select Edges 4, 6, 8, 12, 84, and 87–89 only.

Piezoresistor

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

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

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

4 On the object **ige2**, select Boundary 9 only.

Connections

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

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

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

4 On the object **ige2**, select Boundaries 3, 5, 6, 8, 13–15, and 18 only.

Membrane

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

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

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

4 Locate the **Box Limits** section. In the **x minimum** text field, type `-501`.

5 In the **x maximum** text field, type `501`.

6 In the **y minimum** text field, type `-30`.

7 In the **y maximum** text field, type `1000`.

8 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Model Boundaries

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

2 In the **Settings** window for **Union Selection**, type `Model Boundaries` 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, in the **Selections to add** list, choose **Connections** and **Membrane**.

6 Click **OK**.

Electric currents

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

2 In the **Settings** window for **Union Selection**, type `Electric currents` 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, in the **Selections to add** list, choose **Piezoresistor** and **Connections**.

6 Click **OK**.

Fixed Edges

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

2 In the **Settings** window for **Adjacent Selection**, type `Fixed Edges` in the **Label** text field.

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

4 Click **Add**.

5 In the **Add** dialog, select **Membrane** in the **Input selections** list.

6 Click **OK**.

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

8 From the **Geometric entity level** list, choose **Adjacent edges**.

Fixed Boundaries

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

2 In the **Settings** window for **Difference Selection**, type `Fixed Boundaries` 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 the **Add** button for **Selections to add**.

5 In the **Add** dialog, select **Model Boundaries** in the **Selections to add** list.

6 Click **OK**.

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

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, select **Membrane** in the **Selections to subtract** list.

10 Click **OK**.