

Piezoresistive Pressure Sensor, Shell

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 manufactured by 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](#)).

Note: This model requires the MEMS Module and the Structural Mechanics 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 XducerTM)¹ and part of its associated interconnects are visible. The geometry is shown in [Figure 1](#).

The piezoresistor is assumed to have a uniform p-type dopant density of $1.32 \times 10^{19} \text{ cm}^{-3}$ and a thickness of 400 nm. The interconnects are assumed to have the same thickness but a dopant density of $1.45 \times 10^{20} \text{ cm}^{-3}$. Only a part of the interconnects is included in the geometry because their conductivity is sufficiently high that they do not contribute to the voltage output of the device (in practice, the interconnects would also be thicker in addition to having a higher conductivity but this also has little effect on the solution).

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

1. XducerTM 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.

crystal. In the COMSOL Multiphysics model, a coordinate system rotated 45° about the global Z-axis is added to define the orientation of the crystal.

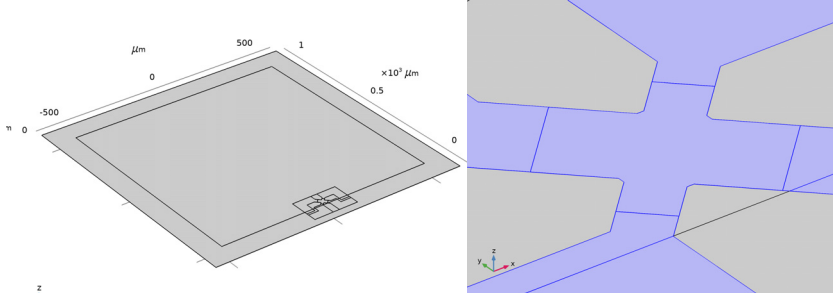


Figure 1: Left: Model geometry. Right: Detail showing the piezoresistor geometry.

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, $\mathbf{E}$, and the current, $\mathbf{J}$, within a piezoresistor is:

$$\mathbf{E} = \rho \cdot \mathbf{J} + \Delta\rho \cdot \mathbf{J} \quad (1)$$

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

$$\Delta\rho = \Pi \cdot \sigma \quad (2)$$

where Π is the piezoresistance tensor (SI units: $\text{Pa}^{-1}\Omega\text{m}$), a material property. Note that the COMSOL Multiphysics definition of Π 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 tensors, 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:

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

The $\Delta\rho$ vector computed from Equation 3 is assembled into matrix form in the following manner in Equation 1:

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

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 XducerTM transducer. An applied voltage (typically 3 V (Ref. 2)) across the [100] orientated arm of the X produces a current (typically 6 mA (Ref. 2)) down this arm. Shear stresses are present in the XducerTM 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.

The Piezoresistance interfaces available with the MEMS Module solve Equation 3 and an inverse form of Equation 4, together with the equations of structural mechanics. In this model the *Piezoresistance, Boundary Currents* interface is used to model the structural equations on the domain level and to solve the electrical equations on a thin layer coincident with a boundary in the model geometry.

Results and Discussion

Figure 2 shows the displacement of the diaphragm as a result of a 100 kPa pressure difference applied to the membrane, at its center the displacement is 1.2 μm . A simple isotropic model for the deform displacement given in Ref. 1 predicts an order of magnitude value for the displacement 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 co-ordinates 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 3, which is 35 MPa. Theoretically the shear stress should be maximal at the midpoint of the edge of the diaphragm. Figure 4 shows the shear stress along the edge in the model. This shows a maximum magnitude of 38 MPa at the center of each of the two edges along which the plot is made.

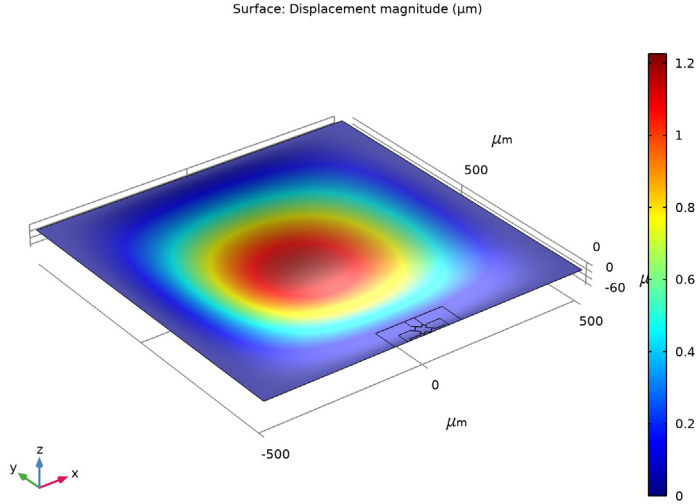


Figure 2: Diaphragm displacement as a result of a 100 kPa applied pressure.

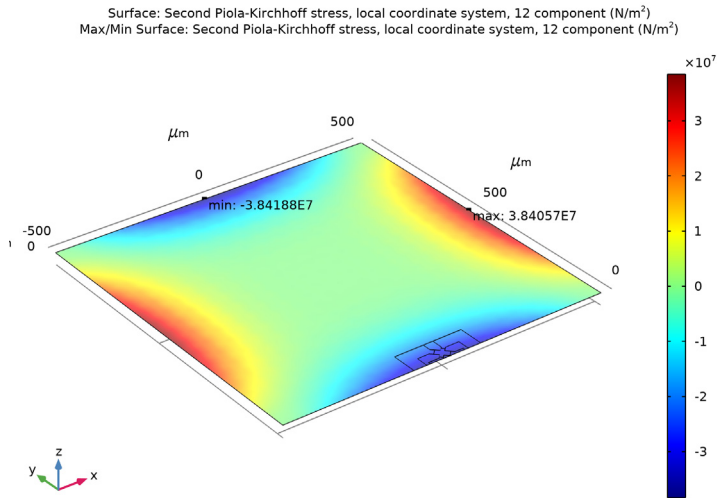


Figure 3: Shear stress, shown in the local coordinate system of the piezoresistor (rotated 45° about the z-axis of the global system). The shear stress has its highest magnitude close to the piezoresistor with a value of approximately -35 MPa.

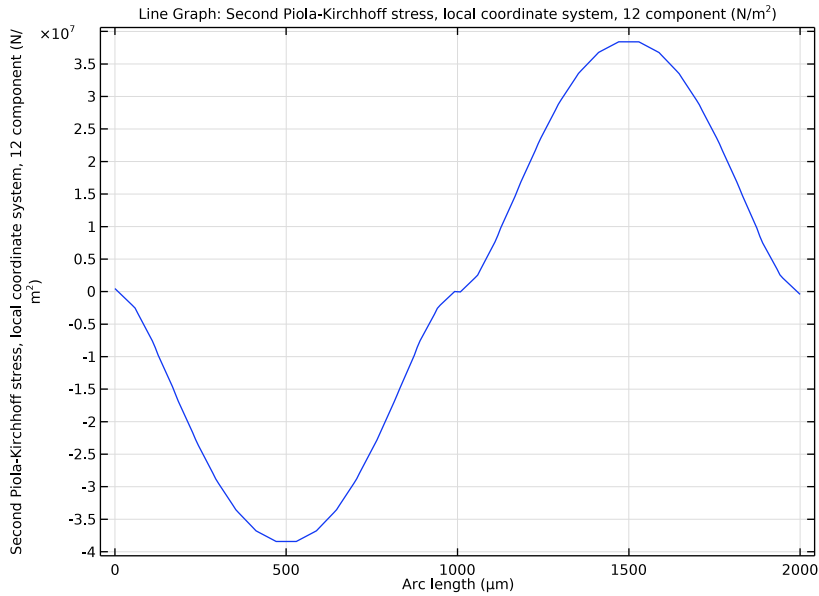


Figure 4: Plot of the local shear stress along two edges of the diaphragm.

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 (c.f. the current quoted in [Ref. 2](#) of 6 mA). The model produces an output voltage of 54 mV, similar to the actual device output of 60 mV quoted in [Ref. 2](#). The detailed current and voltage distribution within the XducerTM is shown in [Figure 5](#). There is clear evidence of the current flow 'spreading out' into the sense electrodes (which are narrower), a phenomena 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.

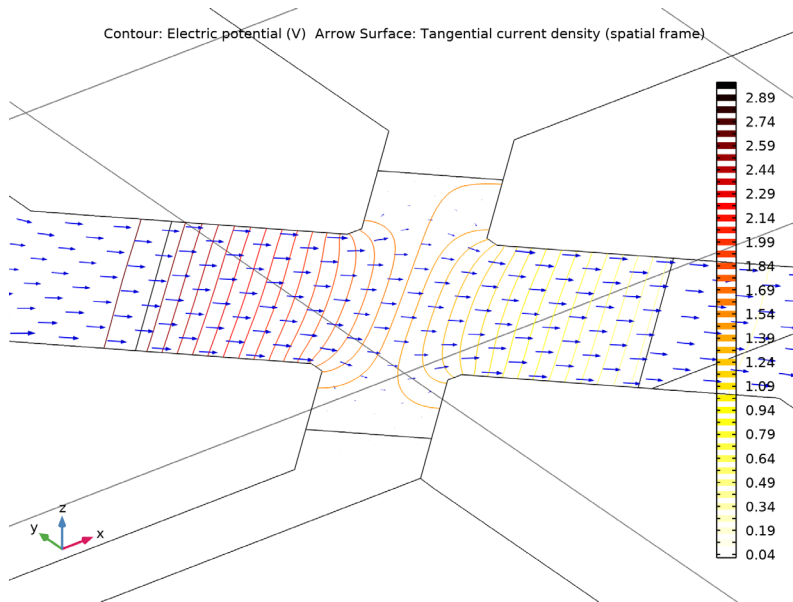


Figure 5: Arrows: Current density, Contours: Electric Potential, for a device driven by a 3 V bias with an applied pressure of 100 kPa.

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 <http://www.freescale.com>).
3. M. Bao, *Analysis and Design Principles of MEMS Devices*, Elsevier B. V., 2005.

Application Library path: MEMS_Module/Sensors/
piezoresistive_pressure_sensor_shell

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, Shell**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies>Stationary**.
- 6 Click  **Done**.

GEOMETRY I

- 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 .

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

- 4 In the **Geometry** toolbar, click  **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**.

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

DEFINITIONS

Piezoresistor

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type **Piezoresistor** in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 9 only.

Connections

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

- 2 In the **Settings** window for **Explicit**, type Connections in the **Label** text field.
- 3 Locate the **Input Entities** section. Click  **Clear Selection**.
- 4 Select Boundaries 3, 5, 6, 8, 13–15, and 18 only.

Membrane

- 1 In the **Definitions** toolbar, click  **Box**.
- 2 In the **Settings** window for **Box**, 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 **Definitions** toolbar, click  **Union**.
- 2 In the **Settings** window for **Union**, 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. Under **Selections to add**, click  **Add**.
- 5 In the **Add** dialog box, in the **Selections to add** list, choose **Connections** and **Membrane**.
- 6 Click **OK**.

Electric currents

- 1 Right-click **Model boundaries** and choose **Duplicate**.
- 2 In the **Settings** window for **Union**, type Electric currents in the **Label** text field.
- 3 Locate the **Input Entities** section. In the **Selections to add** list, select **Membrane**.
- 4 Under **Selections to add**, click  **Delete**.
- 5 Under **Selections to add**, click  **Add**.
- 6 In the **Add** dialog box, select **Piezoresistor** in the **Selections to add** list.
- 7 Click **OK**.

Fixed Edges

- 1 In the **Definitions** toolbar, click  **Adjacent**.
- 2 In the **Settings** window for **Adjacent**, type Fixed Edges in the **Label** text field.

- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Under **Input selections**, click  **Add**.
- 5 In the **Add** dialog box, select **Membrane** in the **Input selections** list.
- 6 Click **OK**.
- 7 In the **Settings** window for **Adjacent**, locate the **Output Entities** section.
- 8 From the **Geometric entity level** list, choose **Adjacent edges**.

Fixed Boundaries

- 1 In the **Definitions** toolbar, click  **Difference**.
- 2 In the **Settings** window for **Difference**, locate the **Geometric Entity Level** section.
- 3 From the **Level** list, choose **Boundary**.
- 4 Locate the **Input Entities** section. Under **Selections to add**, click  **Add**.
- 5 In the **Add** dialog box, select **Model boundaries** in the **Selections to add** list.
- 6 Click **OK**.
- 7 In the **Settings** window for **Difference**, locate the **Input Entities** section.
- 8 Under **Selections to subtract**, click  **Add**.
- 9 In the **Add** dialog box, select **Membrane** in the **Selections to subtract** list.
- 10 Click **OK**.
- 11 In the **Settings** window for **Difference**, type **Fixed Boundaries** in the **Label** text field.

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 (Z-X-Z)** subsection. In the α text field, type **-45[deg]**.

ADD MATERIAL

- 1 In the **Home** toolbar, click  **Add Material** to open the **Add Material** window.
- 2 Go to the **Add Material** window.
- 3 In the tree, select **Piezoresistivity>n-Silicon (single-crystal, lightly doped)**.
- 4 Click **Add to Component** in the window toolbar.
- 5 In the tree, select **Piezoresistivity>p-Silicon (single-crystal, lightly doped)**.
- 6 Click **Add to Component** in the window toolbar.
- 7 In the **Home** toolbar, click  **Add Material** to close the **Add Material** window.

MATERIALS

p-Silicon (single-crystal, lightly doped) (mat2)

- 1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 2 From the **Selection** list, choose **Electric currents**.

SHELL (SHELL)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell (shell)**.
- 2 In the **Settings** window for **Shell**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Model boundaries**.

Thickness and Offset 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Thickness and Offset 1**.
- 2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.
- 3 In the d text field, type 20[um].

Linear Elastic Material 1

- 1 In the **Model Builder** window, click **Linear Elastic Material 1**.
- 2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.
- 3 From the **Solid model** list, choose **Anisotropic**.
- 4 From the **Material data ordering** list, choose **Voigt (11, 22, 33, 23, 13, 12)**.

Shell Local System 1

- 1 In the **Model Builder** window, expand the **Linear Elastic Material 1** node, then click **Shell Local System 1**.
- 2 In the **Settings** window for **Shell Local System**, locate the **Coordinate System Selection** section.
- 3 From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

Fixed Constraint 1

- 1 In the **Physics** toolbar, click  **Edges** and choose **Fixed Constraint**.
- 2 In the **Settings** window for **Fixed Constraint**, locate the **Edge Selection** section.
- 3 From the **Selection** list, choose **Fixed Edges**.

Fixed Constraint 2

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

- 2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Fixed Boundaries**.

Face Load I

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Face Load**.
- 2 In the **Settings** window for **Face Load**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Membrane**.
- 4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.
- 5 In the p text field, type 100[kPa].

ELECTRIC CURRENTS, SINGLE LAYER SHELL (ECS)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents, Single Layer Shell (ecs)**.
- 2 In the **Settings** window for **Electric Currents, Single Layer Shell**, locate the **Boundary Selection** section.
- 3 Click  **Clear Selection**.
- 4 From the **Selection** list, choose **Electric currents**.
- 5 Locate the **Shell Thickness** section. In the d_s text field, type 0.4[um].
- 6 Click the  **Show More Options** button in the **Model Builder** toolbar.
- 7 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.
- 8 Click **OK**.

Current Conservation I

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Electric Currents, Single Layer Shell (ecs)** click **Current Conservation 1**.
- 2 In the **Settings** window for **Current Conservation**, click to expand the **Model Inputs** section.
- 3 In the n_d text field, type 1.45e20[1/cm^3].
- 4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

Piezoresistive Material I

- 1 In the **Model Builder** window, click **Piezoresistive Material 1**.
- 2 In the **Settings** window for **Piezoresistive Material**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Piezoresistor**.

- 4 Locate the **Model Input** section. In the n_d text field, type $1.32 \times 10^{19} [1/\text{cm}^3]$.
- 5 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

Ground 1

- 1 In the **Physics** toolbar, click  **Edges** and choose **Ground**.
- 2 Select Edge 70 only.

Terminal 1

- 1 In the **Physics** toolbar, click  **Edges** and choose **Terminal**.
- 2 Select Edges 11 and 13 only.
- 3 In the **Settings** window for **Terminal**, locate the **Terminal** section.
- 4 From the **Terminal type** list, choose **Voltage**.
- 5 In the V_0 text field, type 3.

Terminal 2

- 1 In the **Physics** toolbar, click  **Edges** and choose **Terminal**.
- 2 Select Edge 7 only.

Terminal 3

- 1 In the **Physics** toolbar, click  **Edges** and choose **Terminal**.
- 2 Select Edges 72 and 73 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 **Mesh Settings** section.
- 3 From the **Sequence type** 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** section.
- 3 Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 60.
- 5 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**.
- 2 Right-click **Free Triangular 1** and choose **Move Down**.
- 3 Right-click **Free Triangular 1** and choose **Move Down**.
- 4 In the **Home** toolbar, click  **Compute**.

RESULTS

Stress (shell)

The default plots show the von Mises stress, the undeformed geometry and the electric potential. Now create a plot of the displacement to compare with [Figure 2](#).

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 **Membrane**.

Displacement

- 1 In the **Results** toolbar, click  **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type Displacement in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.

Surface 1

- 1 Right-click **Displacement** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Shell>Displacement>shell.disp - Displacement magnitude - m**.

Deformation 1

- 1 Right-click **Surface 1** and choose **Deformation**.
- 2 In the **Displacement** toolbar, click  **Plot**.

Now create a plot of the shear stress in the local coordinate system of the piezoresistor, to compare with [Figure 3](#).

In-Plane Shear Stress (Local Coordinates)

- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type In-Plane Shear Stress (Local Coordinates) in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.

Surface 1

- 1 Right-click **In-Plane Shear Stress (Local Coordinates)** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Shell>Stress>Second Piola-Kirchhoff stress, local coordinate system - N/m²>shell.S112 - Second Piola-Kirchhoff stress, local coordinate system, I2 component**.

In-Plane Shear Stress (Local Coordinates)

In the **Model Builder** window, click **In-Plane Shear Stress (Local Coordinates)**.

Max/Min Surface 1

- 1 In the **In-Plane Shear Stress (Local Coordinates)** toolbar, click  **More Plots** and choose **Max/Min Surface**.
- 2 In the **Settings** window for **Max/Min Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **shell.S112 - Second Piola-Kirchhoff stress, local coordinate system, 12 component - N/m²**.
- 3 In the **In-Plane Shear Stress (Local Coordinates)** toolbar, click  **Plot**.

TABLE

Go to the **Table** window.

Create a line plot of the shear stress in the local coordinate system of the piezoresistor, to compare with [Figure 4](#).

RESULTS

In-Plane Shear Stress (Local Coordinate System)

- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **In-Plane Shear Stress (Local Coordinate System)** in the **Label** text field.

Line Graph 1

- 1 Right-click **In-Plane Shear Stress (Local Coordinate System)** and choose **Line Graph**.
- 2 Select Edges 6 and 76 only.
- 3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Shell>Stress>Second Piola-Kirchhoff stress, local coordinate system - N/m²>shell.S112 - Second Piola-Kirchhoff stress, local coordinate system, 12 component**.
- 4 In the **In-Plane Shear Stress (Local Coordinate System)** toolbar, click  **Plot**.

Now create a plot of the detailed current and voltage distribution, to compare with [Figure 5](#).

Study 1/Solution 1 (3) (sol1)

In the **Model Builder** window, under **Results>Datasets** right-click **Study 1/Solution 1 (2) (sol1)** and choose **Duplicate**.

Selection

- 1 In the **Model Builder** window, expand the **Study 1/Solution 1 (3) (sol1)** node, then click **Selection**.

- 2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Piezoresistor**.
- 4 Click  **Zoom to Selection**.
- 5 From the **Selection** list, choose **Electric currents**.

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 (3) (sol1)**.

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 **40**.
- 5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Thermal**.
- 6 Select the **Reverse color table** check box.

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, Single Layer Shell>Currents and charge>ecs.tjx,...,ecs.tjz - Tangential current density (spatial frame)**.
- 3 Locate the **Coloring and Style** section. Select the **Scale factor** check box.
- 4 In the associated text field, type **2e-9**.
- 5 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type **3000**.
- 6 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.
- 7 In the **Current and Voltage** toolbar, click  **Plot**.

Finally evaluate the current drawn by the device and the output voltage.

Global Evaluation 1

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

- 2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Electric Currents, Single Layer Shell>Terminals>ecs.I0_1 - Terminal current - A.**
- 3 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Electric Currents, Single Layer Shell>Terminals>ecs.V0_2 - Terminal voltage - V.**
- 4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
ecs.I0_1	mA	Terminal current
ecs.V0_2-ecs.V0_3	mV	Device Output

- 5 Click  **Evaluate.**

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** check box.
- 8 Select the **Layers to the right** check box.
- 9 Select the **Layers on top** check box.

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** check box.
- 9 Select the **Layers to the left** check box.
- 10 Clear the **Layers on bottom** check box.

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** check box.

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** check box.

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 box, 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 box, 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 box, 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  **Add**.
- 5 In the **Add** dialog box, 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  **Add**.
- 9 In the **Add** dialog box, select **Membrane** in the **Selections to subtract** list.
- 10 Click **OK**.