Capacitive Pressure Sensor
**Introduction**

Capacitive pressure sensors are gaining market share over their piezoresistive counterparts since they consume less power, are usually less temperature sensitive and have a lower fundamental noise floor. This model performs an analysis of a hypothetical sensor design discussed in Ref. 1, using the electromechanics interface. The effect of a rather poor choice of packaging solution on the performance of the sensor is also considered. The results emphasize the importance of considering packaging in the MEMS design process.

**Model Definition**

The model geometry is shown in Figure 1. The pressure sensor is part of a silicon die that has been bonded to a metal plate at 70 °C. Since the geometry is symmetric, only a single quadrant of the geometry needs to be included in the model, and it is possible to use symmetry boundary condition.

Figure 1: The model geometry. Left: The symmetric device geometry, with one quadrant highlighted in blue, showing the symmetry planes. Right: In COMSOL only the highlighted quadrant is modeled, and the symmetry boundary condition is used on the cross section walls.

A detailed 2D section through the functional part of the device is shown in Figure 2. A thin membrane is held at a fixed potential of 1 V. The membrane is separated from a ground plane chamber sealed under high vacuum. The sides of the chamber are insulating to prevent a connection between the membrane and the ground plane (for simplicity the insulating layer is not modeled explicitly in the COMSOL model—this approximation has little effect on the results of the study.).
When the pressure outside of the sealed chamber changes, the pressure difference causes the membrane to deflect. The thickness of the air gap now varies across the membrane and its capacitance to ground therefore changes. This capacitance is then monitored by an interfacing circuit, such as the switched capacitor amplifier circuit discussed in Ref. 1.

Thermal stresses are introduced into the structure as a result of the thermal conductivity mismatch between the silicon die and the metal plate, and the elevated temperature used for the bonding process. These stresses change the deformation of the diaphragm in response to applied pressures and alter the response of the sensor. In addition, because the stresses are temperature dependent, they introduce an undesired temperature dependence to the device output.

Initially the sensor is analyzed in the case where there are no packaging stresses. Then the effect of the packaging stress is considered. First, the device response at fixed temperature is evaluated with the additional packaging stress. Finally the temperature dependence of the device response at a fixed applied pressure is assessed.

**Results and Discussion**

*Figure 3* shows the deformation of the membrane when a pressure of 25 kPa is applied to it, in the absence of packaging stresses. *Figure 4* shows the potential on a plane located between the plates. The deformation of the membrane is of the form expected, and results in a nonuniform potential between the plates.
Figure 3: Quadrant deflection when the pressure difference across the membrane is 25 kPa. As expected the deflection is greatest in the center of the membrane.

Figure 4: Electric potential in the air chamber, plotted on a slice between the two plates of the capacitor. The potential has become nonuniform as a result of the pressure-induced deformation of the diaphragm.
Figure 5: Maximum and mean displacement of the membrane as a function of the applied pressure.

Figure 6: Capacitance of the membrane as a function of applied pressure, both with and without the packaging stresses. The linearized zero pressure capacitance variation, taken from Ref. 1, is also shown for comparison.
Figure 5 shows the mean and maximum displacements of the membrane as a function of applied pressure. At an applied pressure of 10 kPa the diaphragm displacement in the center is 0.89 μm. The average displacement of the diaphragm is 0.27 μm. These values are in good agreement with the approximate model given in Ref. 1 (maximum displacement 0.93 μm, average displacement 0.27 μm).

Figure 6 shows that the capacitance of the device increases nonlinearly with applied pressure. The gradient of the curve plotted is a measure of the sensitivity of the sensor. At zero applied pressure the sensitivity of the model (1/4 of the whole sensor) is $7.3 \times 10^{-6}$ pF/Pa (compare to the value of $6.5 \times 10^{-6}$ pF/Pa given in Ref. 1). The device sensitivity is therefore $29 \times 10^{-6}$ pF/Pa (compare to $26 \times 10^{-6}$ pF/Pa. calculated in Ref. 1). Assuming the interfacing electronics use the switched capacitor amplifier circuit presented in Ref. 1 this corresponds to a sensor transfer function of $29 \times 10^{-6}$ pF/Pa (compared to $26 \times 10^{-6}$ pF/Pa from Ref. 1). Using a smaller pressure step to produce the plot improves the agreement leading to a response at the origin of $6.7 \times 10^{-6}$ pF/Pa ($27 \times 10^{-6}$ pF/Pa for the device, corresponding to 27 μV/Pa). The response is nonlinear, so that at 20 kPa the model output is $14.3 \times 10^{-6}$ pF/Pa (device output 57 pF/Pa or 57 μV/Pa). This nonlinear response adds to the complexity of designing the interfacing circuitry. Note that, for comparison with these figures, the circuitry proposed in Ref. 1, has a noise floor corresponding to a capacitance of $17 \times 10^{-6}$ pF, or 0.6 Pa at zero applied pressure (assuming an average of 100 consecutive measurements). This resolution is approximately four times the fundamental sensitivity of the device imposed by mechanical noise from thermal fluctuations.

Next the response of the device is considered when packaging stresses are present in the model. For this part of the discussion it is assumed that the device is operated at 20°C and that the system was stress and displacement free at the bonding temperature (70°C). Figure 7 shows the displacement of the structure at the room temperature operating point, with an applied pressure of 25 kPa. The membrane displacement at its center is shown in Figure 5. The complex interaction between the thermal stresses and the stresses introduced as a result of the applied pressure has resulted in both an initial offset displacement and an increased dependence of the displacement on the pressure.
Figure 7: The displacement of the structure due to an applied pressure of 25 kPa when packaging stresses are also included in the model. Displacements are shown at the operating temperature of 20 °C, and are assumed to be zero at the die bonding temperature of 70 °C.

Figure 8: Temperature dependence of the capacitance of the packaged device. The capacitance varies with temperature as a result of temperature induced changes in the packaging stress within the diaphragm.
The response of the device with the additional packaging stresses is shown in Figure 6. At zero applied pressure the sensitivity of the COMSOL model has increased from $6.5 \times 10^{-6}$ pF/Pa to $10 \times 10^{-6}$ pF/Pa ($40 \times 10^{-6}$ pF/Pa for the entire device). The effect is even more pronounced at a pressure of 20 kPa, where the model that includes thermal stresses shows a pressure sensitivity of $25 \times 10^{-6}$ pF/Pa (100 pF/Pa for the entire device) compared to the unstrained value of $14.3 \times 10^{-6}$ pF/Pa. The sensitivity of the device to pressure has almost doubled. While this effect might seem desirable, an unwanted dependence on temperature has been introduced into the device response. Since the thermal stresses are temperature dependent, the response of the device is also now temperature dependent. The final study in the model assesses this issue.

Figure 8 shows the capacitance of the device, with an applied pressure of 20 kPa, as the temperature is varied. The temperature sensitivity of the model response is given by the gradient of this curve, approximately $3.5 \times 10^{-3}$ pF/K ($14 \times 10^{-3}$ pF/K for the whole device). With a pressure sensitivity of $25 \times 10^{-6}$ pF/Pa at 20 kPa (for a single quadrant of the device) this corresponds to an equivalent pressure of 140 Pa/K in the sensor output. Compared to the unstrained performance of the sensor (0.6 Pa with the circuit proposed in Ref. 1) this number is very large. The model shows the importance of carefully considering the packaging in the MEMS design process.

Reference


Application Library path: MEMS_Module/Sensors/capacitive_pressure_sensor

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>Electromechanics (emi).

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

**GEOMETRY 1**
The geometry is imported from an external file. Since the structure is symmetric, only a quarter of the physical geometry is required.

*Import 1 (imp1)*
1 On the **Home** toolbar, click **Import**.
2 In the **Settings** window for Import, locate the **Import** section.
3 Click **Browse**.
4 Browse to the application’s Application Libraries folder and double-click the file `capacitive_pressure_sensor.mphbin`.
5 Click **Build All Objects**.
6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Add parameters to the model. These will be used subsequently to perform parametric studies.
GLOBAL DEFINITIONS

Parameters
1. On the Home toolbar, click Parameters.
2. In the Settings window for Parameters, locate the Parameters section.
3. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Name</th>
<th>Expression</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>p0</td>
<td>20[kPa]</td>
<td>2E4 Pa</td>
<td>Pressure</td>
</tr>
<tr>
<td>T0</td>
<td>20[degC]</td>
<td>293.2 K</td>
<td>Operating temperature</td>
</tr>
<tr>
<td>Tref</td>
<td>70[degC]</td>
<td>343.2 K</td>
<td>Die Bonding temperature</td>
</tr>
</tbody>
</table>

SI units or their multiples, such as Pa and kPa, as well as non-SI units, such as degrees Celsius can be entered in the COMSOL Desktop enclosed by square brackets.

Next, add a component coupling operator to compute a derived global quantity from the model. These operators can be convenient for results processing and COMSOL’s solvers can also use them during the solution process, for example to include integral quantities in the equation system. Here, an Average operator is added so that the average displacement of the diaphragm can be computed and a point integration is used to make available the displacement of the center point of the diaphragm.

DEFINITIONS

Average 1 (aveop1)
1. On the Definitions toolbar, click Component Couplings and choose Average.
2. In the Settings window for Average, locate the Source Selection section.
3. From the Geometric entity level list, choose Boundary.
4. Select Boundary 12 only.

Integration 1 (intop1)
1. On the Definitions toolbar, click Component Couplings and choose Integration.
2. In the Settings window for Integration, locate the Source Selection section.
3. From the Geometric entity level list, choose Point.
4. Select Point 4 only.

Next, define selections to simplify the set up of materials and physics.

Box 1
1. On the Definitions toolbar, click Box.
2 In the **Settings** window for Box, locate the **Geometric Entity Level** section.
3 From the **Level** list, choose **Boundary**.
4 Locate the **Box Limits** section. In the **x maximum** text field, type **1e-6**.
5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.
6 Right-click **Box 1** and choose **Rename**.
7 In the **Rename Box** dialog box, type **YZ Symmetry Plane** in the **New label** text field.
8 Click **OK**.

**Box 2**
1 On the **Definitions** toolbar, click **Box**.
2 In the **Settings** window for Box, locate the **Geometric Entity Level** section.
3 From the **Level** list, choose **Boundary**.
4 Locate the **Box Limits** section. In the **y maximum** text field, type **1e-6**.
5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.
6 Right-click **Box 2** and choose **Rename**.
7 In the **Rename Box** dialog box, type **XZ Symmetry Plane** in the **New label** text field.
8 Click **OK**.

**Box 3**
1 On the **Definitions** toolbar, click **Box**.
2 In the **Settings** window for Box, locate the **Box Limits** section.
3 In the **z maximum** text field, type **-100e-6**.
4 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.
5 Right-click **Box 3** and choose **Rename**.
6 In the **Rename Box** dialog box, type **Steel Base** in the **New label** text field.
7 Click **OK**.

**Explicit 1**
1 On the **Definitions** toolbar, click **Explicit**.
2 Select Domain 3 only.
3 Right-click **Explicit 1** and choose **Rename**.
4 In the **Rename Explicit** dialog box, type **Cavity** in the **New label** text field.
5 Click OK.

Explicit 2
1 On the Definitions toolbar, click Explicit.
2 In the Settings window for Explicit, locate the Input Entities section.
3 Select the All domains check box.
4 Right-click Explicit 2 and choose Rename.
5 In the Rename Explicit dialog box, type All domains in the New label text field.
6 Click OK.

Difference 1
1 On the Definitions toolbar, click Difference.
2 In the Settings window for Difference, locate the Input Entities section.
3 Under Selections to add, click Add.
4 In the Add dialog box, select All domains in the Selections to add list.
5 Click OK.
6 In the Settings window for Difference, locate the Input Entities section.
7 Under Selections to subtract, click Add.
8 In the Add dialog box, select Cavity in the Selections to subtract list.
9 Click OK.
10 Right-click Difference 1 and choose Rename.
11 In the Rename Difference dialog box, type Linear Elastic in the New label text field.
12 Click OK.

Next, add the physics settings to the model. These include the pressure forces acting on the sensor, the applied sense voltage, and other appropriate boundary conditions.

ELECTROMECHANICS (EMI)
In the Electromechanics interface, use a Linear Elastic Material node to solve the equations of structural mechanics only. The electric field does not penetrate these regions.

Linear Elastic Material 1
1 On the Physics toolbar, click Domains and choose Linear Elastic Material.
2 In the Settings window for Linear Elastic Material, locate the Domain Selection section.
3 From the Selection list, choose Linear Elastic.

   Apply the structural symmetry boundary condition on the symmetry boundaries.
Symmetry 1
1 On the Physics toolbar, click Boundaries and choose Symmetry.
2 In the Settings window for Symmetry, locate the Boundary Selection section.
3 From the Selection list, choose XZ Symmetry Plane.

Symmetry 2
1 On the Physics toolbar, click Boundaries and choose Symmetry.
2 In the Settings window for Symmetry, locate the Boundary Selection section.
3 From the Selection list, choose YZ Symmetry Plane.
   Note that the electrical symmetry boundary condition (the Zero Charge feature) is applied by default.
   The motion of the structure is constrained in most directions by the structural symmetry boundary conditions. However, the whole device can still slide up and down the z-axis. Apply a point constraint to prevent this.

Prescribed Displacement 2
1 On the Physics toolbar, click Points and choose Prescribed Displacement.
2 Select Point 44 only.
3 In the Settings window for Prescribed Displacement, locate the Prescribed Displacement section.
4 Select the Prescribed in z direction check box.
   Apply a Boundary Load to represent the pressure acting on the surface of the diaphragm.

Boundary Load 1
1 On the Physics toolbar, click Boundaries and choose Boundary Load.
2 Select Boundary 13 only.
3 In the Settings window for Boundary Load, locate the Force section.
4 From the Load type list, choose Pressure.
5 In the p text field, type p0.
   Moving mesh boundary conditions must be applied on boundaries where the air domain deforms and where the default Electromechanical Interface boundary condition does not apply. The Electromechanical Interface boundary condition automatically obtains its selection from the interface between structural and deforming air domains. It applies the appropriate electrical forces to the structural layer and constrains the deformation of the air domain to be equal to that of the structure.
Prescribed Mesh Displacement 1
1 In the Model Builder window, under Component 1 (comp1)>Electromechanics (emi) click Prescribed Mesh Displacement 1.
2 In the Settings window for Prescribed Mesh Displacement, locate the Prescribed Mesh Displacement section.
3 Clear the Prescribed z displacement check box.
   Doing this allows the membrane (and the mesh) to move in the z-direction.
   Add Terminal and Ground features to the model to apply boundary conditions for the electrostatics parts of the problem.

Terminal 1
1 On the Physics toolbar, click Boundaries and choose Terminal.
2 Select Boundary 12 only.
3 In the Settings window for Terminal, locate the Terminal section.
4 From the Terminal type list, choose Voltage.
   The default value of 1 V is fine in this instance.

Ground 1
1 On the Physics toolbar, click Boundaries and choose Ground.
2 Select Boundary 9 only.
   The pressure sensor consists of a silicon die with an enclosed cavity held at a low pressure. The pressure sensor is bonded onto a cylindrical steel plate during the packaging process. COMSOL includes a Material Library with many predefined material properties. This model uses a predefined material for the steel plate, but sets up the silicon as a user-defined material with isotropic material parameters to allow comparison with Ref. 1. The cavity also needs ‘material’ properties (to define the relative permittivity) and a user defined material is used to set the relative permittivity to 1 in this region.

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

Material 1 (mat1)
1 In the Settings window for Material, locate the Material Contents section.
In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Property</th>
<th>Name</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Relative permittivity</td>
<td>epsilonr</td>
<td>11.7</td>
<td>l</td>
<td>Basic</td>
</tr>
<tr>
<td>Young's modulus</td>
<td>E</td>
<td>170 [GPa]</td>
<td>Pa</td>
<td>Basic</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>nu</td>
<td>0.06</td>
<td>l</td>
<td>Basic</td>
</tr>
<tr>
<td>Density</td>
<td>rho</td>
<td>2330</td>
<td>kg/m$^3$</td>
<td>Basic</td>
</tr>
</tbody>
</table>

Right-click Component 1 (comp1)>Materials>Material 1 (mat1) and choose Rename.

In the Rename Material dialog box, type Silicon in the New label text field.

Click OK.

By default, the silicon is in all domains. Some of these selections will be overridden as other materials are added.

Material 2 (mat2)

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

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

From the Selection list, choose Cavity.

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

<table>
<thead>
<tr>
<th>Property</th>
<th>Name</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Relative permittivity</td>
<td>epsilonr</td>
<td>1</td>
<td>l</td>
<td>Basic</td>
</tr>
</tbody>
</table>

Click to expand the Material properties section. Locate the Material Properties section.

From the Material type list, choose Nonsolid.

Right-click Component 1 (comp1)>Materials>Material 2 (mat2) and choose Rename.

In the Rename Material dialog box, type Vacuum in the New label text field.

Click OK.

ADD MATERIAL

On the Home toolbar, click Add Material to open the Add Material window.

Go to the Add Material window.

In the tree, select Built-In>Steel AISI 4340.

Click Add to Component in the window toolbar.

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Relative permittivity</td>
<td>epsilonr</td>
<td>1</td>
<td>l</td>
</tr>
</tbody>
</table>
5 On the Home toolbar, click Add Material to close the Add Material window.

MATERIALS

Steel AISI 4340 (mat3)
1 In the Model Builder window, under Component 1 (comp1)>Materials click Steel AISI 4340 (mat3).
2 In the Settings window for Material, locate the Geometric Entity Selection section.
3 From the Selection list, choose Steel Base.

Next set up a structured mesh to solve the problem on.

MESH 1
In the Model Builder window, under Component 1 (comp1) right-click Mesh 1 and choose Edit Physics-Induced Sequence.

Size
Disable the default free tetrahedral mesh.

Free Tetrahedral 1
1 In the Model Builder window, under Component 1 (comp1) right-click Free Tetrahedral 1 and choose Disable.

Set a maximum element size on the sensor diaphragm.

Size 1
1 Right-click Mesh 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. Select the Maximum element size check box.
5 In the associated text field, type 50e-6.
6 Locate the Geometric Entity Selection section. From the Geometric entity level list, choose Boundary.
7 Select Boundary 3 only.

Create a mapped mesh on the lower surface of the device.

Mapped 1
1 Right-click Mesh 1 and choose More Operations>Mapped.
2 Select Boundaries 3, 16, and 32 only.
In the **Settings** window for Mapped, click **Build All**.

Sweep the surface mesh through the structure.

**Swept 1**

1. Right-click **Mesh 1** and choose **Swept**.
2. In the **Settings** window for Swept, click **Build All**.

Set up a study that sweeps over a range of applied pressures, so that the response of the sensor can be assessed.

**STUDY 1**

**Step 1: Stationary**

1. In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.
2. In the **Settings** window for Stationary, click to expand the **Study extensions** section.
3. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
4. Click **Add**.
   The continuation parameter $p_0$ (**Pressure**) is added by default. This is the correct parameter to sweep over.
5. Click **Range**.
6. In the **Range** dialog box, type 0 in the **Start** text field.
7 In the **Step** text field, type 5000.
8 In the **Stop** text field, type 25000.
9 Click **Add**.
10 On the **Home** toolbar, click **Compute**.

**RESULTS**

*Displacement (emi)*

Much of the structure is not displaced in this initial study. To facilitate results analysis, add a selection to the solution. This will ensure that only the domains of interest are displayed in the plots.

*Study 1/Solution 1 (sol1)*

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/Solution 1 (sol1)**.

**Selection**

1 On the **Results** toolbar, click **Selection**.
2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domains 3 and 4 only.
Displacement (emi)

1. Click the **Zoom Extents** button on the **Graphics** toolbar.

The plot now shows the displacement of the diaphragm only, which, as expected, is maximum in the center of the sensor.

Next, plot the electric potential in an \( xy \)-orientated plane between the sensor diaphragm and the ground plane.

**Slice 1**

1. In the **Model Builder** window, expand the **Potential (emi)** node, then click **Slice 1**.

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

3. From the **Plane** list, choose **xy-planes**.

4. In the **Planes** text field, type 1.

5. Select the **Interactive** check box.

6. In the **Shift** text field, type \(-5.8E-6\).
7 On the Potential (emi) toolbar, click Plot.

Due to the deformation of the diaphragm the potential is non-uniformly distributed in the plane.

Next, plot the deformation of the diaphragm as a function of the pressure differential across it. Include both average and maximum displacements.

1D Plot Group 3
On the Home toolbar, click Add Plot Group and choose 1D Plot Group.

Global 1
1 On the 1D Plot Group 3 toolbar, click Global.

Use the point integration and surface average operators defined earlier to evaluate the displacement at the mid-point of the membrane and the average displacement.

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

3 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Expression</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>intop1(w)</td>
<td>um</td>
<td>Maximum Displacement</td>
</tr>
<tr>
<td>aveop1(w)</td>
<td>um</td>
<td>Average Displacement</td>
</tr>
</tbody>
</table>
ID Plot Group 3
1 In the Model Builder window, under Results click ID Plot Group 3.
2 In the Settings window for 1D Plot Group, click to expand the Title section.
3 From the Title type list, choose Manual.
4 In the Title text area, type Diaphragm displacement.
5 Locate the Plot Settings section. Select the x-axis label check box.
6 In the associated text field, type Pressure (Pa).
7 Select the y-axis label check box.
8 In the associated text field, type Displacement (\mu m).
9 Click to expand the Legend section. From the Position list, choose Lower left.
10 Right-click Results>1D Plot Group 3 and choose Rename.
11 In the Rename 1D Plot Group dialog box, type Diaphragm Displacement vs Pressure in the New label text field.
12 Click OK.
13 On the Diaphragm Displacement vs Pressure toolbar, click Plot.

At an applied pressure of 10 kPa the diaphragm displacement in the centre is 0.89 \mu m. The average displacement of the diaphragm is 0.27 \mu m. These values are in good agreement with the approximate model given in Ref. 1 (maximum displacement 0.93
um, average displacement 0.27 um).

Now plot the sensor capacitance as a function of the applied pressure. If the switched capacitor amplifier described in Ref. 1 is used to produce the output, the sensor output or transfer function is directly proportional to the change in capacitance.

**ID Plot Group 4**

On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.

**Global 1**

1. On the **ID Plot Group 4** toolbar, click **Global**.

   Since the **Terminal** boundary condition was used for the underside of the diaphragm, COMSOL automatically computes its capacitance with respect to ground. The value of the capacitance is available as a variable in results analysis.

2. In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Electromechanics>Terminals>emi.C11 - Capacitance**.

   Next, compare the computed capacitance with the small-displacement, linearized analytic expression derived in Ref. 1.

3. Locate the **y-Axis Data** section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Expression</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>emi.C11</td>
<td>pF</td>
<td>Capacitance</td>
</tr>
<tr>
<td>0.738[pF]*(1+8.87e-6[1/Pa]*p0)</td>
<td>pF</td>
<td>Linearized Analytic Capacitance</td>
</tr>
</tbody>
</table>

**ID Plot Group 4**

1. In the **Model Builder** window, under **Results** click **ID Plot Group 4**.

2. In the **Settings** window for 1D Plot Group, locate the **Title** section.

3. From the **Title type** list, choose **Manual**.

4. In the **Title** text area, type **Model Capacitance vs Pressure**.

5. Locate the **Plot Settings** section. Select the **x-axis label** check box.

6. In the associated text field, type **Pressure (Pa)**.

7. Select the **y-axis label** check box.

8. In the associated text field, type **Capacitance (pF)**.

9. Locate the **Legend** section. From the **Position** list, choose **Upper left**.

10. Right-click **Results>ID Plot Group 4** and choose **Rename**.
11 In the **Rename 1D Plot Group** dialog box, type **Model Capacitance vs Pressure** in the **New label** text field.

12 Click **OK**.

13 On the **Model Capacitance vs Pressure** toolbar, click **Plot**.

The capacitance of the sensor increases with applied pressure. The gradient of the curve plotted gives a useful measure of the response of the device. At the origin, the response of the model (1/4 of the whole sensor) is $7e-6$ pF/Pa, compared to the analytical response of $6.5e-6$ pF/Pa. The response for the whole sensor is $29e-6$ pF/Pa compared to the analytic value of $26e-6$ pF/Pa. With the measurement circuit proposed in Ref. 1 this corresponds to a sensor transfer function of $29 \frac{\mu V}{Pa}$ for the COMSOL model and $26 \frac{\mu V}{Pa}$ for the simple analytic model. The response is nonlinear, so that at 20 kPa the model output is $14e-6$ pF/Pa (device output $57$ pF/Pa).

Next, add thermal expansion to the model to assess the effects of packaging stresses on the device performance.

**ELECTROMECHANICS (EMI)**

**Linear Elastic Material 1**

In the **Model Builder** window, under **Component 1 (comp1)>Electromechanics (emi)** click **Linear Elastic Material 1**.
1 On the **Physics** toolbar, click **Attributes** and choose **Thermal Expansion**.

The model temperature should be set to the previously defined room temperature parameter, $T_0$.

2 In the **Settings** window for Thermal Expansion, locate the **Model Inputs** section.

3 In the $T$ text field, type $T_0$.

   The reference temperature indicates the temperature at which the structure had no thermal strains. In this case, set it to the previously defined parameter, $T_{\text{ref}}$, which represents the temperature at which the silicon die was bonded to the metal carrier plate.

4 Locate the **Thermal Expansion Properties** section. In the $T_{\text{ref}}$ text field, type $T_{\text{ref}}$.

   The user defined properties you added previously for silicon did not include its thermal expansivity, so this must be added.

**MATERIALS**

*Silicon (mat1)*

COMSOL shows a warning in the material properties settings to indicate a missing property.

1 In the table, add a value for the thermal expansivity of silicon to the appropriate row:

   Add a new study to compute the system response including thermal expansivity effects.

**ADD STUDY**

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

**STUDY 2**

*Step 1: Stationary*

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

2 In the **Settings** window for Stationary, locate the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.
4 Click **Add**.

The continuation parameter $p_0$ (Differential Pressure) is added by default. This is the correct parameter to sweep over.

5 Click **Range**.

6 In the **Range** dialog box, type 0 in the **Start** text field.

7 In the **Step** text field, type 5000.

8 In the **Stop** text field, type 25000.

9 Click **Add**.

10 On the **Home** toolbar, click **Compute**.

**RESULTS**

*Displacement (emi)* 1

Create a mirrored dataset to visualize a cross section of the device.

*Mirror 3D* 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.

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

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

*Displacement (emi)* 1

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

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

3 From the **Data set** list, choose **Mirror 3D 1**.
4 On the **Displacement (emi)** toolbar, click **Plot**.

Notice that the entire structure is now displaced at room temperature as a result of thermal expansion.

Now look at the effect of the thermal stress on the response of the sensor.

Add an additional **Global** node to the previously defined plot. This separate node can point to a different data set, enabling a plot of the displacement of the thermally stressed device alongside the unstressed plot.

**Global 1**
In the **Model Builder** window, under **Results>Diaphragm Displacement vs Pressure** right-click **Global 1** and choose **Duplicate**.

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

2 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

Note that the `aveop1(w)` expression has been removed from the table.
On the **Diaphragm Displacement vs Pressure** toolbar, click **Plot**.

The maximum displacement of the membrane is now non-zero at zero applied pressure, as a result of the packaging stress. The gradient of the displacement-pressure line has also changed.

**Model Capacitance vs Pressure**

Now add the thermally stressed results to the Capacitance vs Pressure plot.

1. In the **Model Builder** window, under **Results** click **Model Capacitance vs Pressure**.

**Global 2**

1. On the **Model Capacitance vs Pressure** toolbar, click **Global**.

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

3. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4. Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Electromechanics>Terminals>emi.C11 - Capacitance**.

5. Locate the **y-Axis Data** section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Expression</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>emi.C11</td>
<td>pF</td>
<td>Capacitance with Package Stress</td>
</tr>
</tbody>
</table>
On the **Model Capacitance vs Pressure** toolbar, click **Plot**.

The packaging stress causes a significant change in the response of the device. At zero applied pressure the sensitivity of the COMSOL model has increased to 10e-6 pF/Pa (40e-6 pF/Pa for the entire device). Compare to the unstressed value of 6.5e-6 pf/Pa (29e-6 pF/Pa for the entire device). The effect is even more pronounced at a pressure of 20 kPa, where the model that includes thermal stresses shows a pressure sensitivity of 25e-6 pf/Pa (100 pF/Pa for the entire device), compared to the unstressed pressure sensitivity of 14.3e-6 pf/Pa (sensor output 57 pF/Pa).

![Model Capacitance vs Pressure](image)

It may be possible to calibrate the device to remove the effect of the packaging strains. However, the addition of the thermal stresses to the system has created an additional issue, since the response of the sensor has now become temperature dependent - due to the temperature sensitivity of the thermal strains. This effect is assessed in the final study.

**ADD STUDY**

1. On the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add Study** window.
3. Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.
4. Click **Add Study** in the window toolbar.
5. On the **Home** toolbar, click **Add Study** to close the **Add Study** window.
**STUDY 3**

*Step 1: Stationary*

1. In the Model Builder window, under Study 3 click Step 1: Stationary.
2. In the Settings window for Stationary, locate the Study Extensions section.
3. Select the Auxiliary sweep check box.
   
   Sweep over operating temperature at constant applied pressure, to assess the temperature sensitivity of the device.
4. Click Add.
5. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Parameter name</th>
<th>Parameter value list</th>
<th>Parameter unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>T0</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

6. Click Range.
7. In the Range dialog box, type 290 in the Start text field.
8. In the Step text field, type 5.
9. In the Stop text field, type 300.
10. Click Add.

   For this study disable the default plots, as these will be very similar to those already generated by Study 2.

11. In the Model Builder window, click Study 3.
12. In the Settings window for Study, locate the Study Settings section.
13. Clear the Generate default plots check box.

   Add a plot to show how the sensor response varies with temperature. The response is computed at an applied pressure set by the value of the parameter \( p_0 \), defined as 20 kPa.

**RESULTS**

*1D Plot Group 7*

1. On the Home toolbar, click Add Plot Group and choose 1D Plot Group.
2. In the Settings window for 1D Plot Group, locate the Data section.
3. From the Data set list, choose Study 3/Solution 3 (sol3).
Global 1

1 On the 1D Plot Group 7 toolbar, click Global.

2 In the Settings window for Global, click Replace Expression in the upper-right corner of the y-axis data section. From the menu, choose Component 1>Electromechanics>Terminals>emi.C11 - Capacitance.

3 Locate the y-Axis Data section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Expression</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>emi.C11</td>
<td>pF</td>
<td>Capacitance</td>
</tr>
</tbody>
</table>

1D Plot Group 7

1 In the Model Builder window, under Results click 1D Plot Group 7.

2 In the Settings window for 1D Plot Group, locate the Title section.

3 From the Title type list, choose Manual.

4 In the Title text area, type Model Capacitance vs Operating Temperature.

5 Locate the Plot Settings section. Select the x-axis label check box.

6 In the associated text field, type Operating Temperature (K).

7 Right-click Results>1D Plot Group 7 and choose Rename.

8 In the Rename 1D Plot Group dialog box, type Capacitance vs Operating Temperature in the New label text field.

9 Click OK.
10 On the Capacitance vs Operating Temperature toolbar, click Plot.

At a pressure of 20 kPa the temperature sensitivity of the model is given by the gradient of this curve, approximately 3.5e-3 pF/K (14e-4 pF/K for the whole device). Given the pressure sensitivity of 25e-6 pF/Pa at 20 kPa this corresponds to equivalent pressure of 140 Pa/K in the sensor output. Compared to the noise floor of the measuring circuit proposed in Ref. 1 (0.6 Pa) this number is very large. This model shows that a naive choice of packaging can have a highly detrimental effect on sensor performance.