MEMS Module

Application Library Manual

5.3

COMSOL

MEMS Module Application Library Manual

Version: COMSOL 5.3

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM020902

Created in COMSOL Multiphysics 5.3

Stationary Analysis of a Biased Resonator—2D

Introduction

Silicon micromechanical resonators have long been used for designing sensors and are now becoming increasingly important as oscillators in the consumer electronics market. In this sequence of models, a surface micromachined MEMS resonator, designed as part of a micromechanical filter, is analyzed in detail. The resonator is based on that developed in Ref. 1.

This model performs a stationary analysis of the resonator, with an applied DC bias. It is used as a basis for all the subsequent analyses.

Model Definition

Ref. 1 describes a polysilicon resonator, which is manufactured through a surface micromachining process. The details of this process are outlined in the Stationary Analysis of a Biased Resonator—3D documentation. For this 2D study, a simplified version of the 3D geometry is considered. For simplicity, the resonator is modeled as a 2 µm thick rectangular beam with a length of 45 µm. A Fixed Constraint boundary is applied to each end of the resonator to act as the anchor points at which the resonator is attached to the substrate wafer. The wafer substrate is not explicitly modeled, instead only a 0.1985 µm thick air gap between the resonator and the substrate is included. The effects of the driving electrode are included using Electric Potential boundary conditions applied directly to the underside of the air gap, as shown in Figure 1.

Note that although the structure has a plane of symmetry, which vertically bisects the device, we do not use a symmetry boundary condition. A subsequent model considers the normal modes of the structure, and a symmetry condition eliminates the anti-symmetric modes from this analysis.

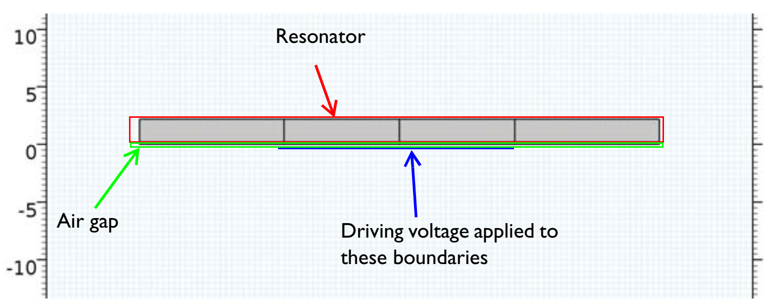


Figure 1: The model geometry. The rectangular resonator and the air gap are highlighted in red and green outline, respectively. The boundaries to which the driving voltage is applied are

highlighted in blue. Note that the vertical dividing lines are not part of the physical geometry of the resonator, but are included to allow a suitable swept mesh to be easily created.

In operation the bottom surface of the silicon resonator is grounded and a driving electrode applies an electric potential to the central portion of the air gap, as shown in Figure 1. Typically a DC bias of 35 V is applied in normal operation of the device. In this model the deformation of the structure is computed with the applied DC bias.

ELECTROMECHANICAL FORCES

Within a vacuum or other medium, forces between charged bodies can be computed on the assumption that a fictitious state of stress exists within the field. The Electromagnetic or Maxwell stress tensor can be used to compute the induced stresses in a material as a result of an electric field as well as surface forces acting on bodies in air or vacuum. Within a material, COMSOL Multiphysics uses the following form of the stress tensor $T_{\mathrm{EM},S}$, which is appropriate for isotropic materials (Ref. 2):

$$T_{EM,S} = -\frac{1}{2}(\mathbf{E}\cdot\mathbf{D} + a_2\mathbf{E}\cdot\mathbf{E})I + \mathbf{E}\mathbf{D}^T + \frac{1}{2}(a_2 - a_1)\mathbf{E}\mathbf{E}^T$$

where $\mathbf{E}$ is the electric field, $\mathbf{D}$ is the electric displacement field, I is the identity tensor, and ε_0 is the permittivity of free space and a_1 and a_2 are material parameters that specify the electrostrictive properties of the material (for this device, assume $a_1 = a_2 = 0$ because the field is in any case very low within the material). This additional stress is applied to the material by the electromechanical solid node. Note that mechanical stresses are usually induced in the material as a result of the net forces acting on the surfaces, in addition to the stress induced by the electric field.

The forces on the surfaces of a solid body can be computed by applying a similar stress term within the vacuum of the form:

$$T_{\mathrm{EM},V} = -\frac{1}{2}(\mathbf{E}\cdot\mathbf{D})I + \mathbf{E}\mathbf{D}^T$$

A net force on the surface typically results from the discontinuity of the stress tensor at the interface. However, since it is undesirable to apply a stress term throughout the vacuum, the force is only available on the surface of solid bodies, via the electromechanical interface node. The surface force is given by:

$$\mathbf{n}_1 T_{\mathrm{EM},V} = -\left(\frac{1}{2}\mathbf{E}\cdot\mathbf{D}\right)\mathbf{n}_1 + (\mathbf{n}_1\cdot\mathbf{E})\mathbf{D}$$

where $\mathbf{n}_1$ is the surface normal, pointing out from the mechanical body.

Results and Discussion

Figure 2 shows the y displacement of the structure with an applied DC bias. As expected the structural displacement is maximal at the center of the geometry. The maximum displacement is 11.2 nm.

The electric potential contours are shown in Figure 3. The fringing fields extend approximately 1 μm into the gap either side of the driving electrode. Note that the fringing fields are not well resolved due to the structure of the swept mesh. In order to investigate these fields the mesh must be refined on either side of the electrode. Also note that the surface of the resonator is assumed to be perfectly grounded. This is a result of the potential boundary condition used and is equivalent to the assumption that the silicon is a perfect conductor. Although the author's of Ref. 1 do not explicitly give the doping in the polysilicon, it is likely that this assumption is relatively poor given the estimated depletion region width of approximately 0.7 μm that is quoted. This model could be extended to include the effects of semiconductor transport and an improvement on this assumption could be made by adding the electric currents which are induced inside the resonator.

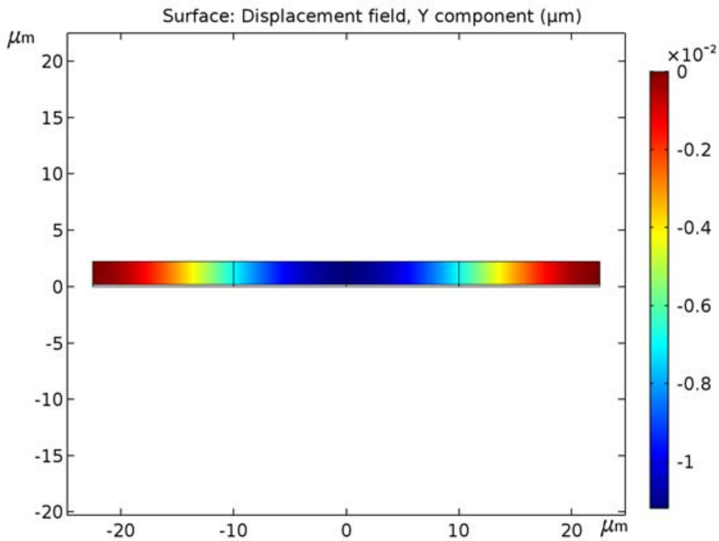


Figure 2: The y-displacement of the resonator as a function of position. The maximum displacement occurs in the center of the resonator, immediately over the biasing electrode.

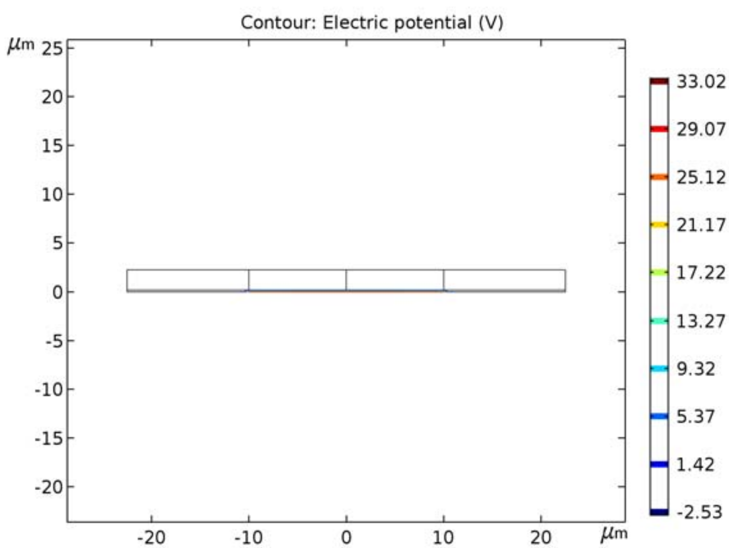


Figure 3: Electric potential contours in the gap between the grounded resonator and the biased driving electrode.

References

1. F.D. Bannon III, J.R. Clark, and C.T.-C. Nguyen, “High-Q HF Microelectromechanical Filters,” *IEEE Journal of Solid State Circuits,* vol. 35, no. 4, pp. 512–526, 2000.

2. J.A. Stratton, *Electromagnetic Theory*, McGraw-Hill, New York, 1941.

Application Library path: `MEMS_Module/Actuators/biased_resonator_2d_basic`

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 **2D**.

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

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**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `22.5`.

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

5 Locate the **Position** section. In the **x** text field, type `-22.5`.

6 In the **y** text field, type `0.1985`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `22.5`.

4 In the **Height** text field, type `0.1985`.

5 Locate the **Position** section. In the **x** text field, type `-22.5`.

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `10`.

4 In the **Height** text field, type `2.1985`.

5 Locate the **Position** section. In the **x** text field, type `-10`.

Mirror 1 (mir1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Click the **Select Box** button on the **Graphics** toolbar.

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

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

5 Select the **Keep input objects** check box.

6 Click **Build All Objects**.

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

Add materials to the model.

ADD MATERIAL

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

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

3 In the tree, select **MEMS>Semiconductors>Si - Polycrystalline Silicon**.

4 Click **Add to Component** in the window toolbar.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Air**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Air (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat2)**.

2 Select Domains 1, 3, 5, and 7 only.

Set the air material property to be non-solid, to ensure the interface solves the electrostatic equation in the spatial frame.

3 In the **Settings** window for **Material**, click to expand the **Material properties** section.

4 Locate the **Material Properties** section. From the **Material type** list, choose **Nonsolid**.

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

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:

Name	Expression	Value	Description
Vdc	35[V]	35 V	DC bias voltage

Set up the solid mechanics and electrostatics boundary conditions.

Add a **Linear Elastic Material** feature to the resonator.

ELECTROMECHANICS (EMI)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromechanics (emi)** and choose **Linear Elastic Material**.

2 Select Domains 2, 4, 6, and 8 only.

Apply a **Fixed Constraint** to both ends of the resonator.

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Electromechanics (emi)** and choose the boundary condition **Structural>Fixed Constraint**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `3 22` in the **Selection** text field.

5 Click **OK**.

Apply the **Ground** condition to the bottom edge of the resonator.

Ground 1

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Electrical> Ground**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `4 9 14 19` in the **Selection** text field.

5 Click **OK**.

Set the bias voltage on the driving electrode with the **Electric Potential** feature.

Electric Potential 1

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Electrical> Electric Potential**.

2 In the **Settings** window for **Electric Potential**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `7 12` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

7 In the V_0 text field, type `Vdc`.

Modify the default mesh settings to suit the model geometry.

A mapped mesh allows good resolution of the small air gap between the driving electrode and the resonator.

MESH 1

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `5 10 15 20` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Distribution**, locate the **Distribution** section.

8 In the **Number of elements** text field, type `15`.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1 3` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Distribution**, locate the **Distribution** section.

7 In the **Number of elements** text field, type `10`.

8 Click **Build All**.

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

STUDY 1

1 In the **Model Builder** window, right-click **Study 1** and choose **Rename**.

2 In the **Rename Study** dialog box, type `Stationary` in the **New label** text field.

3 Click **OK**.

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

RESULTS

Displacement (emi)

1 In the **Model Builder** window, under **Results** right-click **Displacement (emi)** and choose **Rename**.

2 In the **Rename 2D Plot Group** dialog box, type `Biased Displacement` in the **New label** text field.

3 Click **OK**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Biased Displacement** node, then click **Surface 1**.

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

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

Biased Displacement

1 In the **Model Builder** window, under **Results** click **Biased Displacement**.

2 On the **Biased Displacement** toolbar, click **Plot**.

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

Compare the resulting plot with that in .

Create a plot to show the electric potential contours in the gap.

Contour 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Model Builder** window, right-click **2D Plot Group 3** and choose **Contour**.

3 In the **Settings** window for **Contour**, locate the **Levels** section.

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

2D Plot Group 3

Compare the resulting plot with that in .

1 In the **Model Builder** window, under **Results** click **2D Plot Group 3**.

2 On the **2D Plot Group 3** toolbar, click **Plot**.

3 Right-click **Results>2D Plot Group 3** and choose **Rename**.

4 In the **Rename 2D Plot Group** dialog box, type `Electric Potential` in the **New label** text field.

5 Click **OK**.

Created in COMSOL Multiphysics 5.3

Frequency Response of a Biased Resonator—2D

Introduction

Silicon micromechanical resonators have long been used for designing sensors and are now becoming increasingly important as oscillators in the consumer electronics market. In this sequence of models, a surface micromachined MEMS resonator, designed as part of a micromechanical filter, is analyzed in detail. The resonator is based on that developed in Ref. 1.

This model performs a frequency-domain analysis of the structure, which is also biased with its operating DC offset. The analysis begins from the stationary analysis performed in the accompanying model Stationary Analysis of a Biased Resonator—2D; please review this model first.

Model Definition

The geometry, fabrication, and operation of the device are discussed for the Stationary Analysis of a Biased Resonator—2D model.

For the frequency-domain analysis of the structure, consider an applied drive voltage consisting of a 35 V DC offset with a 100 mV drive signal added as a harmonic perturbation. Solve the linearized problem to compute the response of the system.

DAMPING

To obtain the response of the system, you need to add damping to the model. For this study, assume that the damping mechanism is Rayleigh damping or material damping.

To specify the damping, two material constants are required (α_{dM} and β_{dK}). For a system with a single degree of freedom (a mass-spring-damper system) the equation of motion with viscous damping is given by

$$m\frac{d^2u}{dt^2} + c\frac{du}{dt} + ku = f(t)$$

where c is the damping coefficient, m is the mass, k is the spring constant, u is the displacement, t is the time, and $f(t)$ is a driving force.

In the Rayleigh damping model, the parameter c is related to the mass, m, and the stiffness, k, by the equation:

$$c = \alpha_{dM}m + \beta_{dK}k$$

The Rayleigh damping term in COMSOL Multiphysics is proportional to the mass and stiffness matrices and is added to the static weak term.

The damping coefficient, c, is frequently defined as a damping ratio or factor, expressed as a fraction of the critical damping, c_0, for the system such that

$$\xi = \frac{c}{c_0}$$

where for a system with one degree of freedom

$$c_0 = 2\sqrt{km}$$

Finally note that for large values of the quality factor, Q,

$$\xi \cong \frac{1}{2Q}$$

The material parameters α_{dM} and β_{dK} are usually not available in the literature. Often the damping ratio is available, typically expressed as a percentage of the critical damping. It is possible to transform damping factors to Rayleigh damping parameters. The damping factor, ξ, for a specified pair of Rayleigh parameters, α_{dM} and β_{dK}, at the frequency, f, is

$$\xi = \frac{1}{2}\left(\frac{\alpha_{dM}}{2\pi f} + \beta_{dK}2\pi f\right)$$

Using this relationship at two frequencies, f_1 and f_2, with different damping factors, ξ_1 and ξ_2, results in an equation system that can be solved for α_{dM} and β_{dK}:

$$\begin{bmatrix} \frac{1}{4\pi f_1} & \pi f_1 \\ \frac{1}{4\pi f_2} & \pi f_2 \end{bmatrix} \begin{bmatrix} \alpha_{dM} \\ \beta_{dK} \end{bmatrix} = \begin{bmatrix} \xi_1 \\ \xi_2 \end{bmatrix}$$

The damping factors for this model are provided as α_{dM} = 4189 Hz and $\beta_{dK} = 8.29 \cdot 10^{-13}$ s, consistent with the observed Quality factor of 8000 for the fundamental mode.

Results and Discussion

Figure 1 shows the frequency response of the resonator. A clear anti-resonance structure for the frequency response is observable. This response can be compared to that shown in Figure 15 (a) in Ref. 1. Although the experimental results are from a pair of coupled resonators in this instance, the two resonances are sufficiently separate in frequency space that it is possible to distinguish the two modes. If the details of the external circuits were available, a terminal boundary condition with an attached circuit could be used to compute the electrical response of the system for a more direct comparison with the experimental results

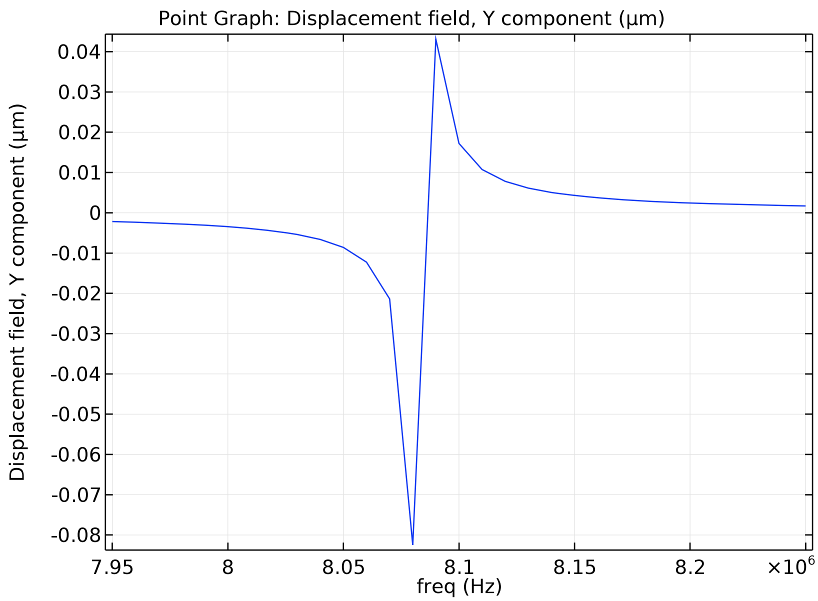


Figure 1: Frequency response of the fundamental mode of the resonator.

Reference

1. F.D. Bannon III, J.R. Clark and C.T.-C. Nguyen, "High-Q HF Microelectromechanical Filters," *IEEE Journal of Solid State Circuits,* vol. 35, no. 4, pp. 512–526, 2000.

Application Library path: `MEMS_Module/Actuators/biased_resonator_2d_freq`

Modeling Instructions

Open the existing stationary study (filename: biased_resonator_2d_basic.mph).

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

Browse to the model's Application Libraries folder and double-click the file `biased_resonator_2d_basic.mph`.

Create parameters for the material damping factors.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, expand the **Global Definitions** node, then click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
alpha	4189[Hz]	4189 Hz	Damping parameter - alpha
beta	8.29e-13[s]	8.29E-13 s	Damping parameter - beta

COMPONENT 1 (COMP1)

In the **Model Builder** window, expand the **Component 1 (comp1)** node.

ELECTROMECHANICS (EMI)

Add damping to the physics settings.

Linear Elastic Material 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Electromechanics (emi)** node.

Damping 1

1 Right-click **Linear Elastic Material 1** and choose **Damping**.

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

3 In the α_{dM} text field, type `alpha`.

4 In the β_{dK} text field, type `beta`.

Add a **Harmonic Perturbation** to the DC bias term, to represent the AC drive voltage.

Harmonic Perturbation 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromechanics (emi)** right-click **Electric Potential 1** and choose **Harmonic Perturbation**.

2 In the **Settings** window for **Harmonic Perturbation**, locate the **Electric Potential** section.

3 In the V_0 text field, type `0.1`.

Set up the frequency domain 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>Prestressed Analysis, Frequency Domain**.

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 2: Frequency-Domain Perturbation

1 In the **Model Builder** window, under **Study 2** click **Step 2: Frequency-Domain Perturbation**.

2 In the **Settings** window for **Frequency-Domain Perturbation**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `range(7.95[MHz],0.01[MHz],8.25[MHz])`.

4 In the **Model Builder** window, right-click **Study 2** and choose **Rename**.

5 In the **Rename Study** dialog box, type `Frequency domain` in the **New label** text field.

6 Click **OK**.

7 In the **Settings** window for **Study**, locate the **Study Settings** section.

8 Clear the **Generate default plots** check box.

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

Produce a plot of the frequency response of the system.

RESULTS

1D Plot Group 4

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 **Frequency domain/Solution 2 (sol2)**.

4 Right-click **1D Plot Group 4** and choose **Rename**.

5 In the **Rename 1D Plot Group** dialog box, type `Frequency Domain` in the **New label** text field.

6 Click **OK**.

Point Graph 1

1 Right-click **Frequency Domain** and choose **Point Graph**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `9` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

7 In the **Expression** text field, type `v`.

8 On the **Frequency Domain** toolbar, click **Plot**.

Compare the resulting plot with .

Created in COMSOL Multiphysics 5.3

Normal Modes of a Biased Resonator—2D

Introduction

Silicon micromechanical resonators have long been used for designing sensors and are now becoming increasingly important as oscillators in the consumer electronics market. In this sequence of models, a surface micromachined MEMS resonator, designed as part of a micromechanical filter, is analyzed in detail. The resonator is based on that developed in Ref. 1.

This model performs a modal analysis on the resonator, with and without an applied DC bias. The analysis begins from the stationary analysis performed in the accompanying model Stationary Analysis of a Biased Resonator—2D; please review this model first.

Model Definition

The geometry, fabrication, and operation of the device are discussed for the "Stationary Analysis of a Biased Resonator" model.

This model performs a modal analysis on the structure, with and without applied DC voltage biases of different magnitudes. The bias already exists as a parameter in the model so the prestressed eigenfrequency solver needs no adjustment to the physics settings. To compute the unbiased eigenfrequency, the solver settings are adjusted to solve only the structural mechanics problem.

Results and Discussion

Figure 1 shows the mode shapes for the resonator under different bias conditions.

The mode shape does not change significantly with applied bias and the first three modes have the expected shapes for a clamped-clamped beam. The frequency of the fundamental is reduced significantly by the applied bias, an effect known as spring softening (the response of higher-order modes was not computed as a function of applied bias).

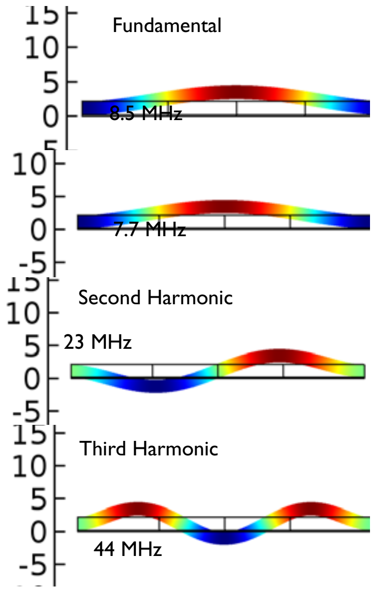


Figure 1: Mode shapes for the resonator under different bias conditions (top line: 0 V; bottom line: 45 V). The mode frequencies are indicated in the figure. The colors visualize the relative y-displacement magnitude.

The spring softening effect can be seen in detail in Figure 2. A clear decrease in the resonant frequency is evident with increasing bias voltage. This figure should be compared with Figure 16 of Ref. 1, where the same effect is apparent.

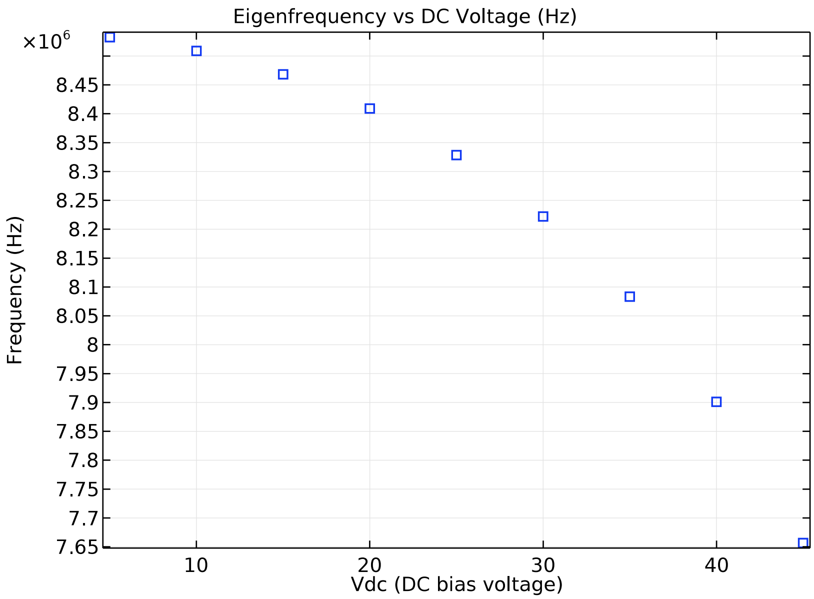


Figure 2: Mode frequency shown against the applied DC voltage bias. The spring softening effect is evident. Compare with Fig. 16 of Ref. 1.

Notes About the COMSOL Implementation

This model excludes certain dependent variables from the solver settings in order to compute the unbiased eigenfrequency. By not computing for the electric potential or the displacement of the air domains, the model is equivalent to a pure solid mechanics problem, solved in the absence of external forces. Excluding dependent variables in the solver in this manner can be useful for debugging models as well as for computing uncoupled problems in this manner.

Reference

1. F.D. Bannon III, J.R. Clark, and C. T.-C. Nguyen, "High-Q HF Microelectromechanical Filters", *IEEE Journal of Solid State Circuits*, vol. 35, no. 4, pp. 512–526, 2000.

Application Library path: `MEMS_Module/Actuators/biased_resonator_2d_modes`

Modeling Instructions

Open the existing stationary study (filename: biased_resonator_2d_basic.mph).

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

Browse to the model's Application Libraries folder and double-click the file `biased_resonator_2d_basic.mph`.

ROOT

Add an unbiased eigenfrequency study. The settings for this study need to be modified so that only the structural part of the problem is solved.

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 **Custom Studies> Eigenfrequency**.

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: Eigenfrequency

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

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

3 Select the **Desired number of eigenfrequencies** check box.

4 In the associated text field, type `3`.

5 In the **Model Builder** window, right-click **Study 2** and choose **Rename**.

6 In the **Rename Study** dialog box, type `Unbiased Eigenfrequency` in the **New label** text field.

7 Click **OK**.

Set up the solver to solve only for the solid mechanics variables.

Solution 2 (sol2)

On the **Study** toolbar, click **Show Default Solver**.

UNBIASED EIGENFREQUENCY

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Dependent Variables 1**.
2 In the **Settings** window for **Dependent Variables**, locate the **General** section.
3 From the **Defined by study step** list, choose **User defined**.
4 In the **Model Builder** window, expand the **Unbiased Eigenfrequency>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** node, then click **Electric potential (comp1.V)**.
5 In the **Settings** window for **Field**, locate the **General** section.
6 Clear the **Solve for this field** check box.
7 Clear the **Store in output** check box.
8 In the **Model Builder** window, under **Unbiased Eigenfrequency>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Spatial coordinates (comp1.xy)**.
9 In the **Settings** window for **Field**, locate the **General** section.
10 Clear the **Solve for this field** check box.
11 Clear the **Store in output** check box.
12 In the **Model Builder** window, click **Unbiased Eigenfrequency**.
13 In the **Settings** window for **Study**, locate the **Study Settings** section.
14 Clear the **Generate default plots** check box.
15 On the **Study** toolbar, click **Compute**.

   Set the data set to be in the material frame for post-processing. This allows the use of the deformation plot attribute.

RESULTS

Unbiased Eigenfrequency/Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Unbiased Eigenfrequency/Solution 2 (sol2)**.
2 In the **Settings** window for **Solution**, locate the **Solution** section.

3 From the **Frame** list, choose **Material (X, Y, Z)**.

Plot the mode shapes.

2D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

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

4 Right-click **2D Plot Group 4** and choose **Rename**.

5 In the **Rename 2D Plot Group** dialog box, type `Unbiased Modes` in the **New label** text field.

6 Click **OK**.

Surface 1

1 Right-click **Unbiased Modes** and choose **Surface**.

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

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

Deformation 1

1 Right-click **Results>Unbiased Modes>Surface 1** and choose **Deformation**.

2 On the **Unbiased Modes** toolbar, click **Plot**.

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

Compare the mode shapes with those shown in for all the modes computed. To switch between the modes click **Unbiased Modes** and choose a different value from the **Eigenfrequency** list.

ROOT

Add a **Prestressed Analysis, Eigenfrequency** 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>Prestressed Analysis, Eigenfrequency**.

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, right-click **Study 3** and choose **Rename**.

2 In the **Rename Study** dialog box, type `Biased Eigenfrequency` in the **New label** text field.

3 Click **OK**.

Create a parametric sweep over DC bias voltage.

Parametric Sweep

On the **Study** toolbar, click **Parametric Sweep**.

BIASED EIGENFREQUENCY

Parametric Sweep

1 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

2 Click **Add**.

3 Click **Range**.

4 In the **Range** dialog box, type `5` in the **Start** text field.

5 In the **Step** text field, type `5`.

6 In the **Stop** text field, type `45`.

7 Click **Add**.

Solve for only the first eigenfrequency.

Step 2: Eigenfrequency

1 In the **Model Builder** window, under **Biased Eigenfrequency** click **Step 2: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** check box.

4 In the associated text field, type `1`.

Disable the default plots.

5 In the **Model Builder** window, click **Biased Eigenfrequency**.

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** check box.

8 On the **Study** toolbar, click **Compute**.

RESULTS

Biased Eigenfrequency/Parametric Solutions 1 (sol5)

1 In the **Model Builder** window, under **Results>Data Sets** click **Biased Eigenfrequency/ Parametric Solutions 1 (sol5)**.
2 In the **Settings** window for **Solution**, locate the **Solution** section.
3 From the **Frame** list, choose **Material (X, Y, Z)**.

Unbiased Modes 1

1 In the **Model Builder** window, under **Results** right-click **Unbiased Modes** and choose **Duplicate**.
2 Right-click **Unbiased Modes 1** and choose **Rename**.
3 In the **Rename 2D Plot Group** dialog box, type `Biased Modes` in the **New label** text field.
4 Click **OK**.
5 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
6 From the **Data set** list, choose **Biased Eigenfrequency/Parametric Solutions 1 (sol5)**.
7 On the **Biased Modes** toolbar, click **Plot**.
8 Click the **Zoom Extents** button on the **Graphics** toolbar.
   Confirm the mode shape is similar to the unbiased fundamental mode.
   Create a plot of eigenfrequency vs. applied DC voltage.

1D Plot Group 6

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 **Biased Eigenfrequency/Parametric Solutions 1 (sol5)**.
4 Right-click **1D Plot Group 6** and choose **Rename**.
5 In the **Rename 1D Plot Group** dialog box, type `Eigenfrequency vs DC voltage` in the **New label** text field.
6 Click **OK**.

Point Graph 1

1 Right-click **Eigenfrequency vs DC voltage** and choose **Point Graph**.
2 In the **Settings** window for **Point Graph**, locate the **Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type `1` in the **Selection** text field.
5 Click **OK**.

6 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

7 In the **Expression** text field, type `emi.freq`.

8 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

9 Click to expand the **Title** section. From the **Title type** list, choose **Custom**.

10 Find the **Type and data** subsection. Clear the **Type** check box.

11 Clear the **Description** check box.

12 Find the **User** subsection. In the **Prefix** text field, type `Eigenfrequency vs DC Voltage`.

13 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

14 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

15 From the **Positioning** list, choose **In data points**.

Compare this plot with that in . Note the spring softening effect.

16 On the **Eigenfrequency vs DC voltage** toolbar, click **Plot**.

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

Created in COMSOL Multiphysics 5.3

Pull-In Voltage for a Biased Resonator—2D

Introduction

Silicon micromechanical resonators have long been used for designing sensors and are now becoming increasingly important as oscillators in the consumer electronics market. In this sequence of models, a surface micromachined MEMS resonator, designed as part of a micromechanical filter, is analyzed in detail. The resonator is based on that developed in Ref. 1.

This model performs a pull-in analysis of the structure, to predict the point at which the biased system becomes unstable. The analysis begins from the stationary analysis performed in the accompanying model Stationary Analysis of a Biased Resonator—2D; please review this model first.

Model Definition

The geometry, fabrication, and operation of the device are discussed for the "Stationary Analysis of a Biased Resonator" model.

This model computes the pull-in voltage for the resonator by solving an inverse problem. The y-coordinate of the resonator midpoint is computed using an integration operator (`intop1`). The inverse problem that COMSOL solves computes the DC voltage that must be applied to the beam in order to move the midpoint to a set y-coordinate, `yset`. This is achieved by adding a global equation for the DC voltage, `VdcSP`, applied to the resonator. The equation `intop1(y)-yset=0` is solved to determine the value of `VdcSP`. This means that `VdcSP` is adjusted until the midpoint of the resonator has a y-coordinate given by the set value, `yset`. Essentially COMSOL is being asked to find the voltage that allows the beam to exist in equilibrium (stable or unstable) at a given displacement. Solving the problem in this manner avoids complications with trying to solve a problem with no solution (which is what happens if the voltage is continuously ramped up eventually exceeding the pull-in voltage). The result of the analysis is a displacement versus voltage plot, with a minimum at the pull-in voltage. Note that for a linear spring, the pull-in displacement corresponds to 1/3 of the gap distance. Although the inclusion of geometric nonlinearities in the solid mechanics solver means that the pull-in displacement changes slightly from this value, it is usually most efficient to search around this point for the pull-in voltage.

Results and Discussion

Figure 1 shows the voltage-displacement curve for the resonator at equilibrium, for y-coordinates that correspond to displacements of around 1/3 of the gap size. The pull-in voltage is 63.3 V.

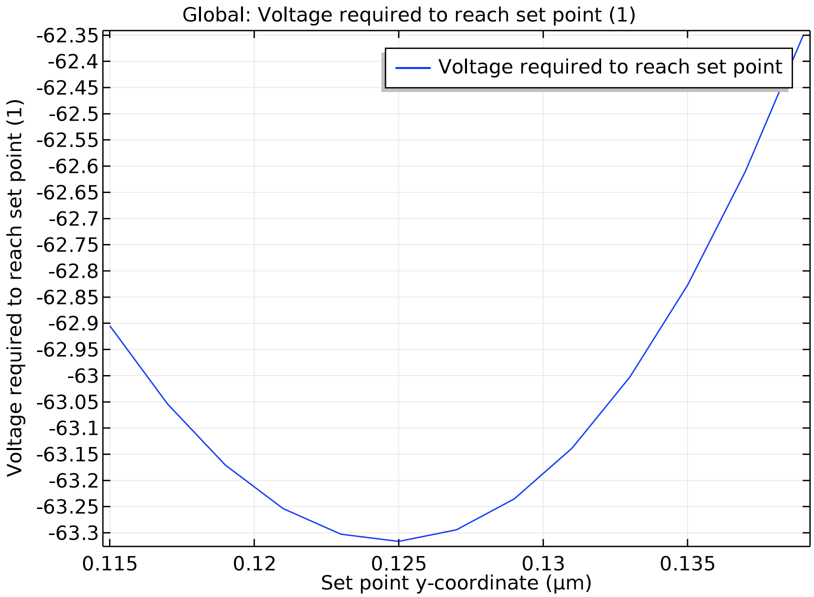


Figure 1: Voltage required to achieve a set displacement versus the target displacement. The pull-in voltage is the minimum of the plot: 63.3 V.

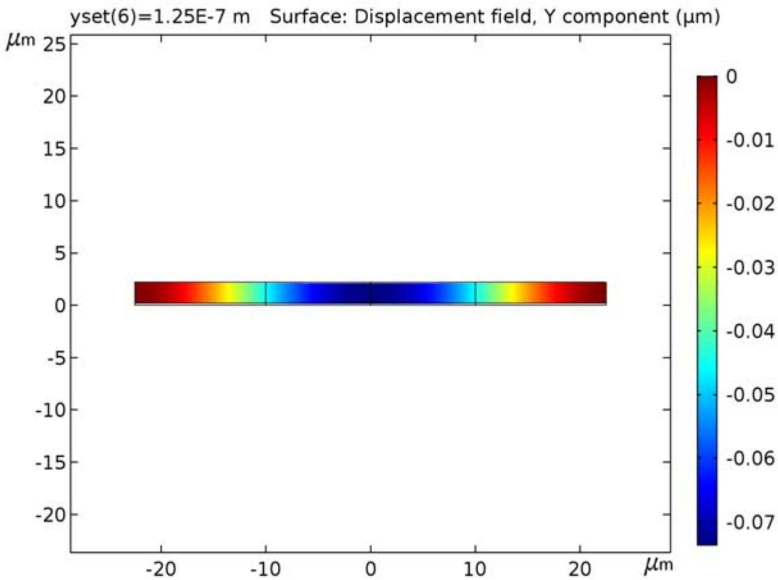


Figure 2: y-displacement of the resonator at pull-in. The displacement at pull-in is 74 nm. For a linear spring the displacement at pull in would be 66 nm.

Figure 2 shows the *y*-displacement of the resonator at the pull-in voltage. The maximum displacement at pull-in is 74 nm. This is comparable to the (approximate) linear spring value of 66 nm.

Notes About the COMSOL Implementation

To compute the voltage required to generate the desired displacement of the beam, use a global equation. A common use of global equations is for computing the value of a dependent variable based on an ordinary differential equation in the dependent variable itself. However, it is also possible to couple a global equation with the other PDEs in the model as a powerful tool to solve certain kinds of inverse problems. This model uses a global equation to compute the potential applied to the drive electrode. The equation takes the form

$$y_0 = y_{\mathrm{set}}$$

where y_0 is the y-coordinate of the midpoint of the beam's underside and y_{set} is the desired y-coordinate. COMSOL Multiphysics computes the voltage to satisfy the constraint implied by the above equation.

Reference

1. F.D. Bannon III, J.R. Clark and C.T.-C. Nguyen, "High-Q HF Microelectromechanical Filters," *IEEE Journal of Solid State Circuits,* vol. 35, no. 4, pp. 512–526, 2000.

Application Library path: `MEMS_Module/Actuators/biased_resonator_2d_pull_in`

Modeling Instructions

Open the existing stationary study (filename: biased_resonator_2d_basic.mph).

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

Browse to the model's Application Libraries folder and double-click the file `biased_resonator_2d_basic.mph`.

Add a parameter to set the y-coordinate of the midpoint of the resonator.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, expand the **Global Definitions** node, then click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
yset	100[nm]	1E-7 m	Set point y-coordinate

Add an integration operator to compute the actual displacement.

DEFINITIONS

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 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type 8 in the **Selection** text field.

6 Click **OK**.

Change the drive potential to the value VdcSP - which will be solved for in a global equation.

ELECTROMECHANICS (EMI)

Electric Potential 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Electromechanics (emi)** node, then click **Electric Potential 1**.

2 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

3 In the V_0 text field, type VdcSP[V].

4 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

Add a global equation to compute the voltage for a given displacement, VdcSP.

Global Equations 1

1 In the **Model Builder** window, right-click **Electromechanics (emi)** and choose **Global> Global Equations**.

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

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
VdcSP	(intop1(y)-yset)/yset	0	0	

Set up a parametric sweep over the displacement set point, yset.

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, right-click **Study 2** and choose **Rename**.

2 In the **Rename Study** dialog box, type `Pull In` in the **New label** text field.

3 Click **OK**.

PULL IN

Step 1: Stationary

1 In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.

2 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
yset	`range(115[nm],2[nm],140[nm])`	

The problem is highly non-linear due to the presence of the global equation, so the solver settings need to be adjusted accordingly.

Solution 2 (sol2)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Pull In>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and termination** section.

5 Locate the **Method and Termination** section. From the **Nonlinear method** list, choose **Automatic highly nonlinear (Newton)**.

6 On the **Study** toolbar, click **Compute**.

Determine the pull-in voltage by plotting `VdcSP` vs. `yset`.

RESULTS

1D Plot Group 6

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

2 In the **Model Builder** window, right-click **1D Plot Group 6** and choose **Rename**.

3 In the **Rename 1D Plot Group** dialog box, type `Pull In Plot` in the **New label** text field.

4 Click **OK**.

Global 1

1 Right-click **Pull In Plot** and choose **Global**.

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

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

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

Expression	Unit	Description
VdcSP	1	Voltage required to reach set point

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

6 In the **Expression** text field, type `yset`.

7 On the **Pull In Plot** toolbar, click **Plot**.

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

Compare the resulting plot with . The pull in voltage is the minimum of the curve: 63.3 V at yset=125 nm.

Produce a plot of the y displacement of the structure at pull-in.

Biased Displacement 1

1 In the **Model Builder** window, under **Results** right-click **Biased Displacement** and choose **Duplicate**.

2 Right-click **Biased Displacement 1** and choose **Rename**.

3 In the **Rename 2D Plot Group** dialog box, type `Pull In Displacement` in the **New label** text field.

4 Click **OK**.

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

6 From the **Data set** list, choose **Pull In/Solution 2 (sol2)**.

7 From the **Parameter value (yset (m))** list, choose **1.25E-7**.

8 On the **Pull In Displacement** toolbar, click **Plot**.

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

Compare the resulting plot with .

Created in COMSOL Multiphysics 5.3

Stationary Analysis of a Biased Resonator—3D

Introduction

Silicon micromechanical resonators have long been used for designing sensors and are now becoming increasingly important as oscillators in the consumer electronics market. This sequence of models analyzes in detail a surface micromachined MEMS resonator, designed as part of a micromechanical filter. The resonator is based on that developed in Ref. 1.

This model performs a stationary analysis of the resonator, with an applied DC bias. It serves as a basis for all the subsequent analyses.

Model Definition

The model consists of a poly-silicon resonator, which is manufactured through a surface micromachining process. Initially, a silicon wafer is coated with 0.75 µm of oxide and 0.15 µm of silicon nitride to isolate the micromachined parts from the wafer ground plane. Polysilicon electrodes with a thickness of 0.3 µm are deposited next. A sacrificial layer of oxide is then deposited to a thickness of 198.5 nm. Note that in Ref. 1 the sacrificial oxide is actually 1.3 µm, but the gap thickness was adjusted to this value for the purposes of simulation to account for the depletion layer in the silicon. This model uses the same adjustment to enable the simulations to be directly compared with those presented in the paper. Holes are etched in the sacrificial layer (to provide anchor points for the resonator) and the structural polysilicon is deposited with a thickness of 1.9 µm.

The structure has a plane of symmetry, so it is possible to model only half of the geometry explicitly, although care must be taken to mirror the geometry before performing a modal analysis. Figure 1 shows the geometry.

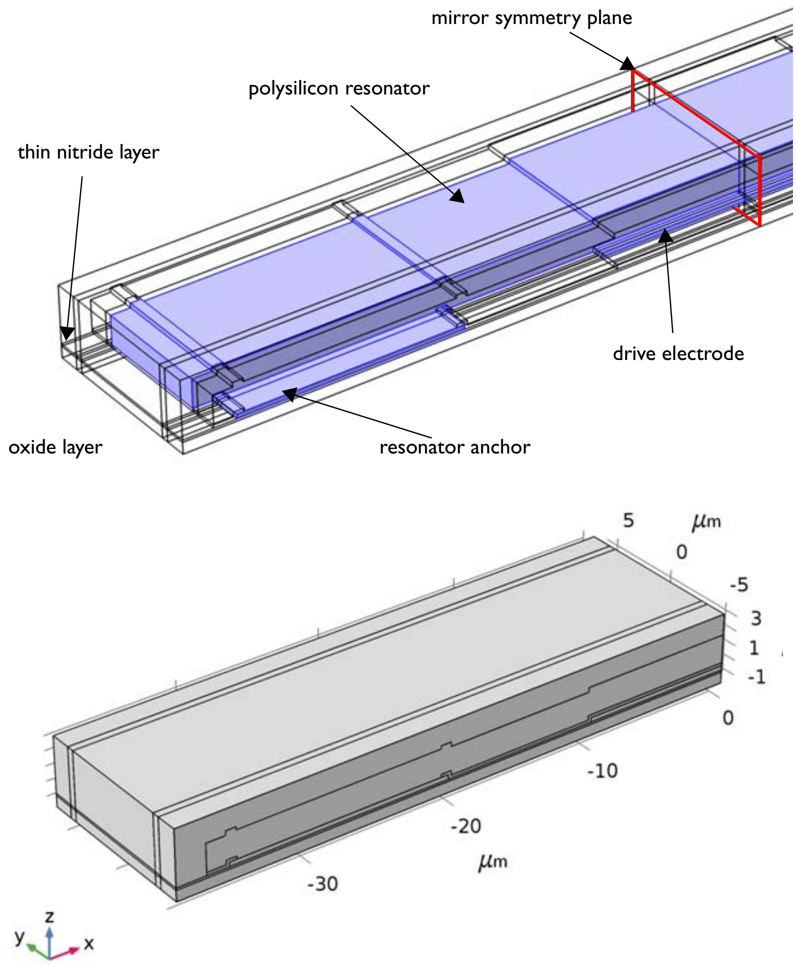


Figure 1: Top: Device geometry. The wafer itself is not shown explicitly, but is represented in the model by a ground plane on the underside of the geometry. Bottom: The model geometry as it appears in COMSOL.

The layers of deposited material from the ground plane up are: silicon oxide, silicon nitride (too thin to see clearly), polysilicon electrodes/air gap (etched sacrificial oxide), polysilicon resonator, and air.

In operation, both the silicon resonator and the underlying wafer are grounded and an electric voltage is applied to the driving electrode, which is bisected by the symmetry plane.

Typically a DC bias of 35 V is applied in normal operation of the device. The assumption is made that the polysilicon is a perfect conductor, so the bias voltage is applied on all exterior surfaces of the resonator and its anchor as a potential boundary condition. In this model, the deformation of the structure is computed with the applied DC bias. Note that the silicon oxide and nitride are assumed to be rigid for the solid mechanics simulations, so the structure is anchored at the base of its electrode, and these domains are not included in the solid mechanics equations.

ELECTROMECHANICAL FORCES

Within a vacuum or other medium, forces between charged bodies can be computed on the assumption that a fictitious state of stress exists within the field. The Electromagnetic or Maxwell stress tensor can be used to compute the induced stresses in a material as a result of an electric field as well as surface forces acting on bodies in air or vacuum. In this model, it is assumed that the polysilicon is doped sufficiently heavily that it can be treated as a perfect conductor. The electric field is assumed to be zero inside the resonator, which means that the Maxwell stress tensor is zero inside the material and there are no volumetric electrical forces. The Maxwell stress tensor in the medium surrounding the resonator, where the electric field is non-zero is (Ref. 2)

$$T_{\mathrm{EM},\,V} = -\frac{1}{2}(\mathbf{E}\cdot\mathbf{D})I + \mathbf{E}\mathbf{D}^{T}$$

A net force on the surface typically results from the discontinuity of the stress tensor at the interface. However, because it is undesirable to apply a stress term throughout the vacuum, the force is only computed on the surface of the resonator, and is applied by the Electromechanical Interface node. The surface force is given by

$$\mathbf{n}_1 T_{\mathrm{EM},\,V} = -\left(\frac{1}{2}\mathbf{E}\cdot\mathbf{D}\right)\mathbf{n}_1 + (\mathbf{n}_1\cdot\mathbf{E})\mathbf{D}$$

where $\mathbf{n}_1$ is the surface normal, pointing out from the mechanical body.

Results and Discussion

Figure 2 shows the z displacement of the structure with an applied DC bias. As expected the structural displacement is maximal on the symmetry plane at the center of the device. The maximum displacement is 13 nm. Electric potential isosurfaces are also shown in Figure 2. As expected, the isobars are uniformly distributed and closest together between the resonator and the electrode. This corresponds to a region of uniform electric field. Around the electrode the fringing fields can also be seen. Note that the surface of the resonator is assumed to be perfectly grounded. This is a result of the potential boundary condition used and is equivalent to the assumption that the polysilicon is a perfect conductor.

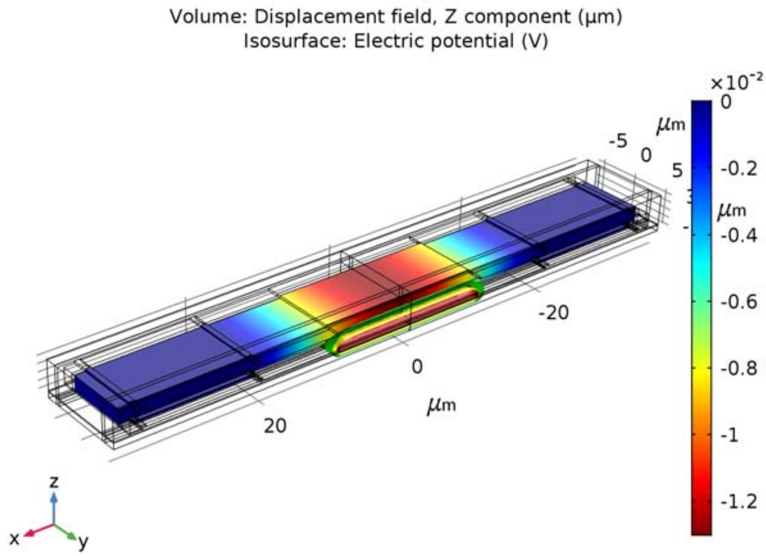


Figure 2: The z-displacement of the resonator as a function of position. The maximum displacement occurs in the center of the resonator, immediately over the biasing electrode. Electric potential isosurfaces with values of 10 V (green), 20 V (yellow), and 30 V (red) are also shown.

References

1. F.D. Bannon III, J.R. Clark, and C.T.-C. Nguyen, "High-Q HF Microelectromechanical Filters," *IEEE Journal of Solid State Circuits,* vol. 35, no. 4, pp. 512–526, 2000.

2. J.A. Stratton, *Electromagnetic Theory*, McGraw-Hill, New York, 1941.

Application Library path: `MEMS_Module/Actuators/biased_resonator_3d_basic`

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**.

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

GEOMETRY 1

1 On the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `biased_resonator_3d_geom_sequence.mph`.

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

Add a parameter for the applied DC bias.

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:

Name	Expression	Value	Description
Vdc	35[V]	35 V	DC bias voltage

Create selections to facilitate easy set up of the boundary conditions.

DEFINITIONS

Explicit 1

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 1** and choose **Rename**.

5 In the **Rename Explicit** dialog box, type `All domains` in the **New label** text field.

6 Click **OK**.

Box 1

1 On the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for **Box**, locate the **Box Limits** section.

3 In the **z minimum** text field, type `-2`.

4 In the **z maximum** text field, type `-1`.

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

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

7 Right-click **Box 1** and choose **Rename**.

8 In the **Rename Box** dialog box, type `Ground Plane` in the **New label** text field.

9 Click **OK**.

Box 2

1 On the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for **Box**, locate the **Box Limits** section.

3 In the **z minimum** text field, type `-1`.

4 In the **z maximum** text field, type `-0.9`.

5 Right-click **Box 2** and choose **Rename**.

6 In the **Rename Box** dialog box, type `Oxide` in the **New label** text field.

7 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 minimum** text field, type `-0.4`.

4 In the **z maximum** text field, type `-0.35`.

5 Right-click **Box 3** and choose **Rename**.

6 In the **Rename Box** dialog box, type `Nitride` in the **New label** text field.

7 Click **OK**.

Box 4

1 On the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for **Box**, locate the **Box Limits** section.

3 In the **x minimum** text field, type `-0.1`.

4 In the **x maximum** text field, type `0.1`.

5 In the **y minimum** text field, type `-4.2`.

6 In the **z minimum** text field, type `-0.15`.

7 In the **z maximum** text field, type `-0.1`.

8 Right-click **Box 4** and choose **Rename**.

9 In the **Rename Box** dialog box, type `Electrode` in the **New label** text field.

10 Click **OK**.

Ball 1

1 On the **Definitions** toolbar, click **Ball/Disk**.

2 In the **Settings** window for **Ball**, locate the **Ball Center** section.

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

4 Locate the **Ball Radius** section. In the **Radius** text field, type `0.1`.

Box 5

1 On the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for **Box**, locate the **Box Limits** section.

3 In the **y maximum** text field, type `4.8`.

4 In the **z minimum** text field, type `-0.35`.

5 In the **z maximum** text field, type `0.05`.

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

Box 6

1 Right-click **Box 5** and choose **Duplicate**.

2 In the **Settings** window for **Box**, locate the **Box Limits** section.

3 In the **x minimum** text field, type `-15`.

4 In the **x maximum** text field, type `15`.

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, In the **Selections to add** list, choose **Ball 1** and **Box 5**.

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 **Box 6** 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 `Resonator` in the **New label** text field.

12 Click **OK**.

Union 1

1 On the **Definitions** toolbar, click **Union**.

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

3 Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, In the **Selections to add** list, choose **Electrode** and **Resonator**.

5 Click **OK**.

6 Right-click **Union 1** and choose **Rename**.

7 In the **Rename Union** dialog box, type `PolySi` in the **New label** text field.

8 Click **OK**.

Difference 2

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, In the **Selections to subtract** list, choose **Oxide**, **Nitride**, and **PolySi**.

9 Click **OK**.

10 Right-click **Difference 2** and choose **Rename**.

11 In the **Rename Difference** dialog box, type `Air` in the **New label** text field.

12 Click **OK**.

Adjacent 1

1 On the **Definitions** toolbar, click **Adjacent**.

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

3 Under **Input selections**, click **Add**.

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

5 Click **OK**.

6 Right-click **Adjacent 1** and choose **Rename**.

7 In the **Rename Adjacent** dialog box, type `Resonator Boundaries` in the **New label** text field.

8 Click **OK**.

Adjacent 2

1 On the **Definitions** toolbar, click **Adjacent**.

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

3 Under **Input selections**, click **Add**.

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

5 Click **OK**.

6 Right-click **Adjacent 2** and choose **Rename**.

7 In the **Rename Adjacent** dialog box, type `Electrode Boundaries` in the **New label** text field.

8 Click **OK**.

Adjacent 3

1 On the **Definitions** toolbar, click **Adjacent**.

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

3 Under **Input selections**, click **Add**.

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

5 Click **OK**.

6 Right-click **Adjacent 3** and choose **Rename**.

7 In the **Rename Adjacent** dialog box, type `Nitride Boundaries` in the **New label** text field.

8 Click **OK**.

Adjacent 4

1 On the **Definitions** toolbar, click **Adjacent**.

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

3 Under **Input selections**, click **Add**.

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

5 Click **OK**.

6 Right-click **Adjacent 4** and choose **Rename**.

7 In the **Rename Adjacent** dialog box, type `Geometry Exterior Boundaries` in the **New label** text field.

8 Click **OK**.

Difference 3

1 On 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 **Resonator 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 **Geometry Exterior Boundaries** in the **Selections to subtract** list.

10 Click **OK**.

11 Right-click **Difference 3** and choose **Rename**.

12 In the **Rename Difference** dialog box, type `Resonator Exterior Boundaries` in the **New label** text field.

13 Click **OK**.

Difference 4

1 On 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 **Electrode 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 **Geometry Exterior Boundaries** in the **Selections to subtract** list.

10 Click **OK**.

11 Right-click **Difference 4** and choose **Rename**.

12 In the **Rename Difference** dialog box, type `Electrode Exterior Boundaries` in the **New label** text field.

13 Click **OK**.

Intersection 1

1 On the **Definitions** toolbar, click **Intersection**.

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

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

4 Locate the **Input Entities** section. Under **Selections to intersect**, click **Add**.

5 In the **Add** dialog box, In the **Selections to intersect** list, choose **Resonator Boundaries** and **Nitride Boundaries**.

6 Click **OK**.

7 Right-click **Intersection 1** and choose **Rename**.

8 In the **Rename Intersection** dialog box, type `Fixed Boundaries` in the **New label** text field.

9 Click **OK**.

Box 7

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 minimum** text field, type `-0.1`.

5 In the **x maximum** text field, type `0.1`.

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

7 Right-click **Box 7** and choose **Rename**.

8 In the **Rename Box** dialog box, type `Symmetry Boundaries` in the **New label** text field.

9 Click **OK**.

MATERIALS

Add materials to the model.

ADD MATERIAL

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

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

3 In the tree, select **MEMS>Semiconductors>Si - Polycrystalline Silicon**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Si - Polycrystalline Silicon (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Si - Polycrystalline Silicon (mat1)**.

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

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

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **MEMS>Insulators>Si3N4 - Silicon nitride**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Si3N4 - Silicon nitride (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Si3N4 - Silicon nitride (mat2)**.

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

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

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **MEMS>Insulators>SiO2 - Silicon oxide**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

SiO2 - Silicon oxide (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **SiO2 - Silicon oxide (mat3)**.

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

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

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Air**.

3 Click **Add to Component** in the window toolbar.

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat4)**.

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

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

Set the air material property to be non-solid, to ensure the interface solves the electrostatics equations in the spatial frame.

4 Click to expand the **Material properties** section. Locate the **Material Properties** section. From the **Material type** list, choose **Nonsolid**.

ELECTROMECHANICS (EMI)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromechanics (emi)** 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 **Resonator**.

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Electromechanics (emi)** and choose the boundary condition **Structural>Fixed Constraint**.

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

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

Symmetry 1

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Structural>Symmetry**.

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

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

Fixed Mesh 2

1 Right-click **Electromechanics (emi)** and choose the domain setting **Deformed Mesh>Fixed Mesh**.

2 In the **Settings** window for **Fixed Mesh**, locate the **Domain Selection** section.

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

Fixed Mesh 3

1 Right-click **Component 1 (comp1)>Electromechanics (emi)>Fixed Mesh 2** and choose **Duplicate**.

2 In the **Settings** window for **Fixed Mesh**, locate the **Domain Selection** section.

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

Fixed Mesh 4

1 Right-click **Component 1 (comp1)>Electromechanics (emi)>Fixed Mesh 3** and choose **Duplicate**.

2 In the **Settings** window for **Fixed Mesh**, locate the **Domain Selection** section.

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

Prescribed Mesh Displacement 2

1 In the **Model Builder** window, right-click **Electromechanics (emi)** and choose the boundary condition **Deformed Mesh>Prescribed Mesh Displacement**.

2 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Boundary Selection** section.

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

4 Locate the **Prescribed Mesh Displacement** section. Clear the **Prescribed y displacement** check box.

5 Clear the **Prescribed z displacement** check box.

Ground 1

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Electrical>Ground**.

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

3 From the **Selection** list, choose **Resonator Exterior Boundaries**.

Ground 2

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Electrical>Ground**.

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

3 From the **Selection** list, choose **Ground Plane**.

Electric Potential 1

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Electrical>Electric Potential**.

2 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

3 In the V_0 text field, type `Vdc`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Electrode Exterior Boundaries**.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.

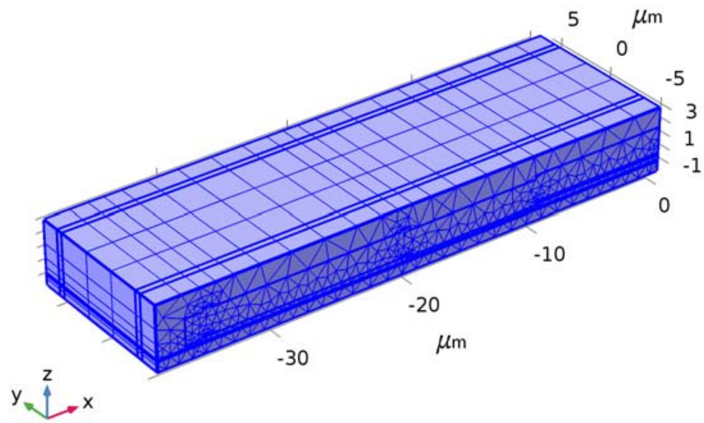
3 From the **Selection** list, choose **Free Triangular Mesh**.

4 Click **Build Selected**.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 In the **Settings** window for **Swept**, click **Build Selected**.



STUDY 1

1 In the **Model Builder** window, right-click **Study 1** and choose **Rename**.

2 In the **Rename Study** dialog box, type `Stationary` in the **New label** text field.

3 Click **OK**.

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

RESULTS

3D Plot Group 3

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

2 On the **Results** toolbar, click **3D Plot Group**.

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

4 From the **Data set** list, choose **Mirror 3D 1**.

Volume 1

1 Right-click **3D Plot Group 3** and choose **Volume**.

2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Electromechanics (Solid Mechanics)>Displacement> Displacement field (material and geometry frames)>w - Displacement field, Z component**.

3 Locate the **Coloring and Style** section. Select the **Reverse color table** check box.

Isosurface 1

1 In the **Model Builder** window, under **Results** right-click **3D Plot Group 3** and choose **Isosurface**.

2 In the **Settings** window for **Isosurface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromechanics (Electrical Quasistatics)>Electric>V - Electric potential**.

3 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

4 In the **Levels** text field, type `10 20 30`.

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

6 Clear the **Color legend** check box.

3D Plot Group 3

1 Right-click **3D Plot Group 3** and choose **Rename**.

2 In the **Rename 3D Plot Group** dialog box, type `Biased Displacement` in the **New label** text field.

3 Click **OK**.

Compare the resulting plot with .

Created in COMSOL Multiphysics 5.3

Frequency Response of a Biased Resonator—3D

Introduction

Silicon micromechanical resonators have long been used for designing sensors and are now becoming increasingly important as oscillators in the consumer electronics market. In this sequence of models, a surface micromachined MEMS resonator, designed as part of a micromechanical filter, is analyzed in detail. The resonator is based on that developed in Ref. 1.

This model performs a frequency-domain analysis of the structure, which is also biased with its operating DC offset. The analysis begins from the stationary analysis performed in the accompanying model Stationary Analysis of a Biased Resonator—3D; please review this model first.

Model Definition

The geometry, fabrication, and operation of the device are discussed for the "Stationary Analysis of a Biased Resonator—3D" model.

For the frequency-domain analysis of the structure, consider an applied drive voltage consisting of a 35 V DC offset with a 100 mV drive signal added as a harmonic perturbation. Solve the linearized problem to compute the response of the system.

In general, for resonant structures like this model, a very fine mesh is required to achieve accurate frequency response results. In the interest of saving time, we choose to use a relatively coarse mesh for this tutorial. As a result the resonant peak will shift if a more refined mesh is used.

DAMPING

To obtain the response of the system, you need to add damping to the model. For this study, assume that the damping mechanism is Rayleigh damping or material damping.

To specify the damping, two material constants are required (α_{dM} and β_{dK}). For a system with a single degree of freedom (a mass-spring-damper system) the equation of motion with viscous damping is given by

$$m\frac{d^2u}{dt^2}+c\frac{du}{dt}+ku = f(t)$$

where c is the damping coefficient, m is the mass, k is the spring constant, u is the displacement, t is the time, and $f(t)$ is a driving force.

In the Rayleigh damping model, the parameter c is related to the mass, m, and the stiffness, k, by the equation:

$$c = \alpha_{dM} m + \beta_{dK} k$$

The Rayleigh damping term in COMSOL Multiphysics is proportional to the mass and stiffness matrices and is added to the static weak term.

The damping coefficient, c, is frequently defined as a damping ratio or factor, expressed as a fraction of the critical damping, c_0, for the system such that

$$\xi = \frac{c}{c_0}$$

where for a system with one degree of freedom

$$c_0 = 2\sqrt{km}$$

Finally note that for large values of the quality factor, Q,

$$\xi \cong \frac{1}{2Q}$$

The material parameters α_{dM} and β_{dK} are usually not available in the literature. Often the damping ratio is available, typically expressed as a percentage of the critical damping. It is possible to transform damping factors to Rayleigh damping parameters. The damping factor, ξ, for a specified pair of Rayleigh parameters, α_{dM} and β_{dK}, at the frequency, f, is

$$\xi = \frac{1}{2}\left(\frac{\alpha_{dM}}{2\pi f} + \beta_{dK} 2\pi f\right)$$

Using this relationship at two frequencies, f_1 and f_2, with different damping factors, ξ_1 and ξ_2, results in an equation system that can be solved for α_{dM} and β_{dK}:

$$\begin{bmatrix} \frac{1}{4\pi f_1} & \pi f_1 \\ \frac{1}{4\pi f_2} & \pi f_2 \end{bmatrix} \begin{bmatrix} \alpha_{dM} \\ \beta_{dK} \end{bmatrix} = \begin{bmatrix} \xi_1 \\ \xi_2 \end{bmatrix}$$

The damping factors for this model are provided as α_{dM} = 4189 Hz and $\beta_{dK} = 8.29 \cdot 10^{-13}$ s, consistent with the observed Quality factor of 8000 for the fundamental mode.

Results and Discussion

Figure 1 shows the frequency response of the resonator. This response can be compared to that shown in Figure 4 in Ref. 1. Although the experimental results in Ref. 1 are from a pair of coupled resonators in this instance, the two resonances are sufficiently separate in frequency space that it is possible to distinguish the two modes. If the details of the external circuits were available, a terminal boundary condition with an attached circuit could be used to compute the electrical response of the system for a more direct comparison with the experimental results.

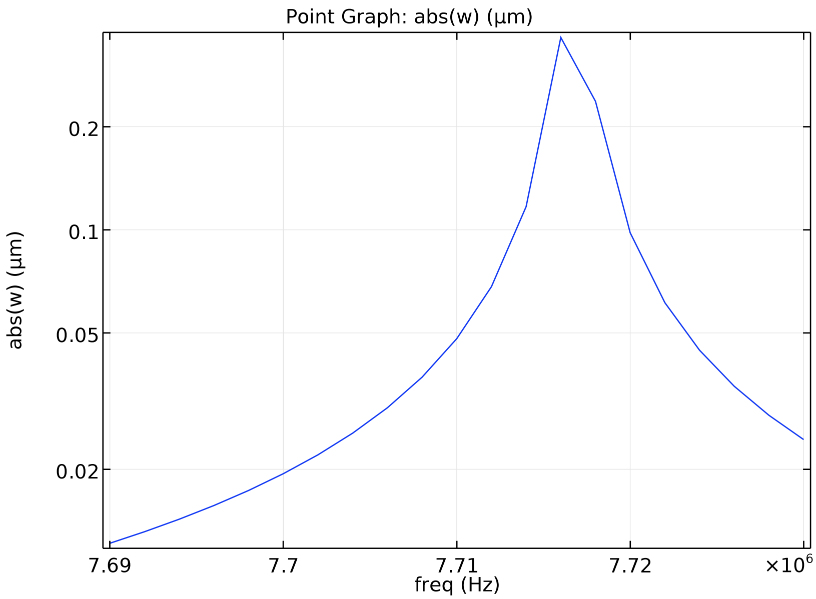


Figure 1: Frequency response of the fundamental mode of the resonator.

Reference

1. F.D. Bannon III, J.R. Clark and C.T.-C. Nguyen, "High-Q HF Microelectromechanical Filters," *IEEE Journal of Solid State Circuits,* vol. 35, no. 4, pp. 512–526, 2000.

Application Library path: `MEMS_Module/Actuators/biased_resonator_3d_freq`

Modeling Instructions

Open the existing stationary study (filename: biased_resonator_3d_basic.mph).

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

Browse to the model's Application Libraries folder and double-click the file `biased_resonator_3d_basic.mph`.

Create parameters for the material damping factors.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, expand the **Global Definitions** node, then click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
Q	8000	8000	Resonator quality factor
f0	8[MHz]	8E6 Hz	Approximate resonance frequency
alpha	4*pi*f0/(3*Q)	4189 Hz	Damping parameter
beta	1/(6*pi*f0*Q)	8.289E-13 s	Damping parameter

Add damping to the physics settings.

COMPONENT 1 (COMP1)

In the **Model Builder** window, expand the **Component 1 (comp1)** node.

ELECTROMECHANICS (EMI)

Linear Elastic Material 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Electromechanics (emi)** node.

Damping 1

1 Right-click **Linear Elastic Material 1** and choose **Damping**.

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

3 In the α_{dM} text field, type `alpha`.

4 In the β_{dK} text field, type `beta`.

Add a **Harmonic Perturbation** to the DC bias term, to represent the offset AC drive voltage.

Harmonic Perturbation 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromechanics (emi)** right-click **Electric Potential 1** and choose **Harmonic Perturbation**.

2 In the **Settings** window for **Harmonic Perturbation**, locate the **Electric Potential** section.

3 In the V_0 text field, type `0.1`.

Set up the frequency domain 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>Prestressed Analysis, Frequency Domain**.

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 2: Frequency-Domain Perturbation

1 In the **Model Builder** window, under **Study 2** click **Step 2: Frequency-Domain Perturbation**.

2 In the **Settings** window for **Frequency-Domain Perturbation**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `range(7.69[MHz],0.002[MHz],7.73[MHz])`.

4 In the **Model Builder** window, right-click **Study 2** and choose **Rename**.

5 In the **Rename Study** dialog box, type `Frequency Domain` in the **New label** text field.

6 Click **OK**.

Disable the default plots.

7 In the **Settings** window for **Study**, locate the **Study Settings** section.

8 Clear the **Generate default plots** check box.

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

Produce a plot of the frequency response of the system.

RESULTS

1D Plot Group 4

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 **Frequency Domain/Solution 2 (sol2)**.

Point Graph 1

1 Right-click **1D Plot Group 4** and choose **Point Graph**.

2 Select Point 254 only.

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

4 In the **Expression** text field, type `abs(w)`.

5 On the **1D Plot Group 4** toolbar, click **Plot**.

6 Click the **y-Axis Log Scale** button on the **Graphics** toolbar.

1D Plot Group 4

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 4** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Frequency Domain` in the **New label** text field.

3 Click **OK**.

Compare the resulting plot with .

Created in COMSOL Multiphysics 5.3

Normal Modes of a Biased Resonator—3D

Introduction

Silicon micromechanical resonators have long been used for designing sensors and are now becoming increasingly important as oscillators in the consumer electronics market. In this sequence of models, a surface micromachined MEMS resonator, designed as part of a micromechanical filter, is analyzed in detail. The resonator is based on that developed in Ref. 1.

This model performs a modal analysis on the resonator, with and without an applied DC bias. The analysis begins from the stationary analysis performed in the accompanying model Stationary Analysis of a Biased Resonator—3D; please review this model first.

Model Definition

The geometry, fabrication, and operation of the device are discussed for the “Stationary Analysis of a Biased Resonator—3D” model. In this example it is no longer possible to model half of the geometry using symmetry boundary conditions, because doing so excludes all the antisymmetric vibrational modes. The geometry is therefore mirrored prior to performing the analyses, as shown in Figure 1. Note that the model could still be solved with the original geometry and symmetry boundary conditions, however the antisymmetric modes would be excluded from the solutions.

This model performs a modal analysis on the structure, with and without applied DC voltage biases of different magnitudes. The bias already exists as a parameter in the model so the prestressed eigenfrequency solver needs no adjustment to the physics settings. To compute the unbiased eigenfrequency, the solver settings are adjusted to solve only the structural mechanics problem.

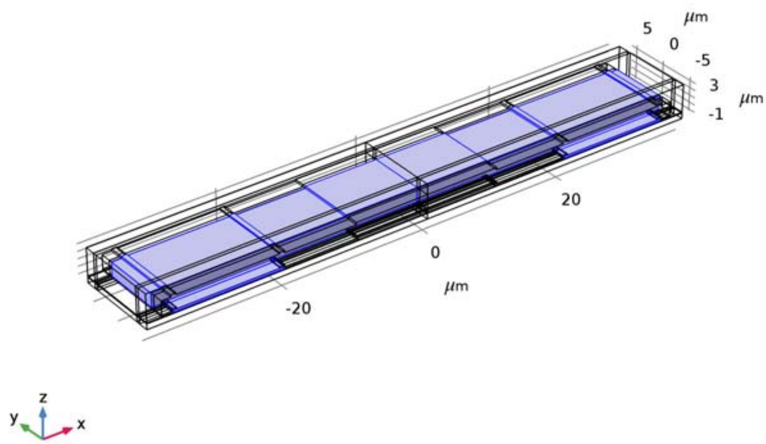


Figure 1: Model geometry. In order to capture the anti-symmetric vibrational modes, it is necessary to mirror the symmetric geometry prior to solving the model. The original symmetry plane is in the center of the geoemtry.The resonator itself is shown highlighted.

Results and Discussion

Figure 2 shows the normal modes of the device, together with the eigenfrequency, in the unbiased state. The lowest three normal modes are symmetric and anti-symmetric bending modes and a torsional mode.

The symmetric bending mode is employed during the operation of the device, and its shape does not change significantly with applied bias. However, the frequency of the mode is reduced significantly by the applied bias, an effect known as spring softening. The spring softening effect can be seen in detail in Figure 2. A clear decrease in the resonant frequency is evident with increasing bias voltage. This figure should be compared with Figure 16 of Ref. 1 which shows measured experimental data for the same device. Data extracted from Ref. 1 is shown in Figure 3 along with the simulation results. The agreement between the model and the data is excellent.

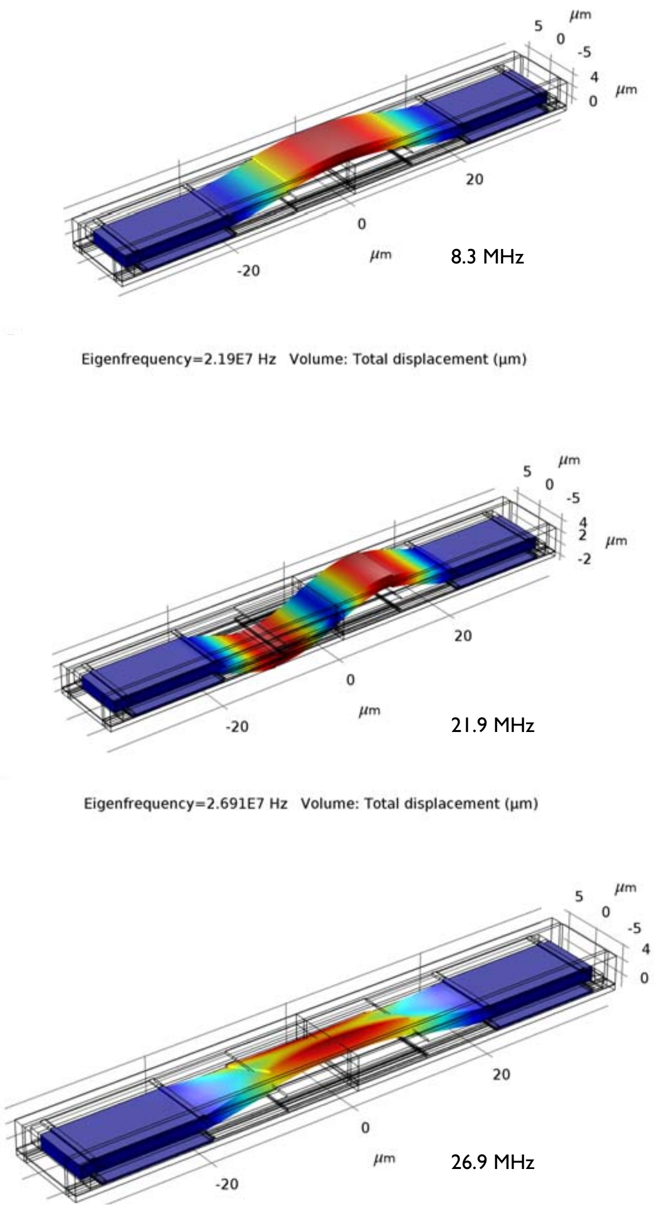


Figure 2: Normal modes of the unbiased device, together with the frequency of the mode.

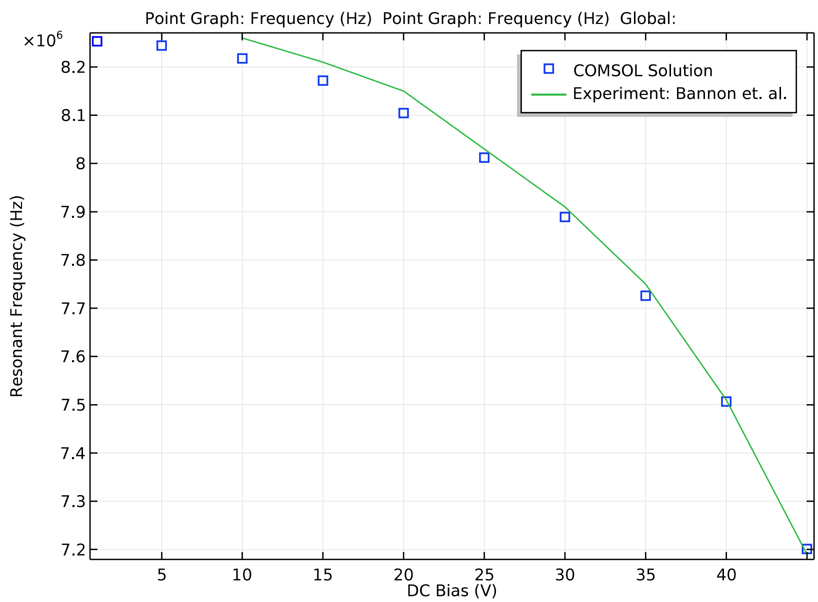


Figure 3: Resonant frequency of the first normal mode (a symmetric bending mode) as a function of applied DC bias. Both the COMSOL simulation data and the experimental data from Ref. 1 *are shown in the plot.*

Reference

1. F.D. Bannon III, J.R. Clark and C.T.-C. Nguyen, "High-Q HF Microelectromechanical Filters," *IEEE Journal of Solid State Circuits*, vol. 35, no. 4, pp. 512–526, 2000.

Application Library path: `MEMS_Module/Actuators/biased_resonator_3d_modes`

Modeling Instructions

ROOT

Open the existing stationary study (filename: biased_resonator_3d_basic.mph).

1 From the **File** menu, choose **Open**.

2 Browse to the model's Application Libraries folder and double-click the file `biased_resonator_3d_basic.mph`.

RESULTS

Biased Displacement

Mirror the geometry so that asymmetric eigenmodes can be modeled.

GEOMETRY 1

Mirror 1 (mir1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 In the **Settings** window for **Mirror**, locate the **Normal Vector to Plane of Reflection** section.

3 In the **z** text field, type `0`.

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

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

6 Locate the **Input** section. Select the **Keep input objects** check box.

7 Click **Build All Objects**.

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

Import experimental data into the model for comparison with the simulation.

DEFINITIONS

Interpolation 1 (int1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.

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

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `biased_resonator_3d_modes_experiment.txt`.

6 Click **Import**.

7 Locate the **Units** section. In the **Arguments** text field, type `Hz`.

8 In the **Function** text field, type `V`.

9 Locate the **Interpolation and Extrapolation** section. From the **Extrapolation** list, choose **Specific value**.

10 In the **Value outside range** text field, type `NaN`.

Disable the symmetry node to allow anti-symmetric nodes.

ELECTROMECHANICS (EMI)

Symmetry 1

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Electromechanics (emi)** node.
2. Right-click **Symmetry 1** and choose **Disable**.

ROOT

Add a study to compute the unbiased vibrational modes.

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 **Custom Studies> Eigenfrequency**.
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: Eigenfrequency

Solve for the first three modes.

1. In the **Model Builder** window, click **Step 1: Eigenfrequency**.
2. In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
3. Select the **Desired number of eigenfrequencies** check box.
4. In the associated text field, type `3`.

   Disable the electric potential and mesh displacement degrees of freedom to solve only the structural problem. This will give the vibrational modes in the absence of an electric field.

Solution 2 (sol2)

1. On the **Study** toolbar, click **Show Default Solver**.
2. In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Dependent Variables 1**.

3 In the **Settings** window for **Dependent Variables**, locate the **General** section.
4 From the **Defined by study step** list, choose **User defined**.
5 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** node, then click **Electric potential (comp1.V)**.
6 In the **Settings** window for **Field**, locate the **General** section.
7 Clear the **Solve for this field** check box.
8 Clear the **Store in output** check box.
9 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Spatial coordinates (comp1.xyz)**.
10 In the **Settings** window for **Field**, locate the **General** section.
11 Clear the **Solve for this field** check box.
12 Clear the **Store in output** check box.
13 In the **Model Builder** window, right-click **Study 2** and choose **Rename**.
14 In the **Rename Study** dialog box, type `Unbiased Eigenfrequency` in the **New label** text field.
15 Click **OK**.
16 In the **Settings** window for **Study**, locate the **Study Settings** section.
17 Clear the **Generate default plots** check box.
18 On the **Study** toolbar, click **Compute**.

Change the data set frame to show results in the material frame. This allows the use of the deformation plot attribute.

RESULTS

Unbiased Eigenfrequency/Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Unbiased Eigenfrequency/Solution 2 (sol2)**.
2 In the **Settings** window for **Solution**, locate the **Solution** section.
3 From the **Frame** list, choose **Material (X, Y, Z)**.

Create a plot that shows the unbiased modes.

3D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

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

Volume 1

1 Right-click **3D Plot Group 4** and choose **Volume**.

2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1> Electromechanics (Solid Mechanics)>Displacement>emi.disp - Total displacement**.

3 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

3D Plot Group 4

1 Right-click **Results>3D Plot Group 4>Volume 1** and choose **Deformation**.

2 In the **Model Builder** window, under **Results** right-click **3D Plot Group 4** and choose **Rename**.

3 In the **Rename 3D Plot Group** dialog box, type `Unbiased Modes` in the **New label** text field.

4 Click **OK**.

Compare the mode shapes with those shown in for all the modes computed. To switch between the modes click **Unbiased Modes** and choose a different value from the **Eigenfrequency** list.

Add a **Prestressed Analysis, Eigenfrequency** 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> Prestressed Analysis, Eigenfrequency**.

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, right-click **Study 3** and choose **Rename**.

2 In the **Rename Study** dialog box, type `Biased Eigenfrequency` in the **New label** text field.

3 Click **OK**.

Create a parametric sweep over DC bias voltage.

Parametric Sweep

On the **Study** toolbar, click **Parametric Sweep**.

BIASED EIGENFREQUENCY

Parametric Sweep

1 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

2 Click **Add**.

3 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Vdc		

4 Click **Range**.

5 In the **Range** dialog box, type 5 in the **Start** text field.

6 In the **Stop** text field, type 45.

7 In the **Step** text field, type 5.

8 Click **Add**.

Solve for only the first eigenfrequency.

Step 2: Eigenfrequency

1 In the **Model Builder** window, under **Biased Eigenfrequency** click **Step 2: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** check box.

4 In the associated text field, type 1.

Disable the default plots.

5 In the **Model Builder** window, click **Biased Eigenfrequency**.

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** check box.

8 On the **Study** toolbar, click **Compute**.

Create a plot of eigenfrequency vs. applied DC voltage.

RESULTS

1D Plot Group 5

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 **Biased Eigenfrequency/Parametric Solutions 1 (sol5)**.

Point Graph 1

1 Right-click **1D Plot Group 5** and choose **Point Graph**.

2 Select Point 1 only.

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

4 In the **Expression** text field, type `emi.freq`.

5 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

6 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

8 From the **Positioning** list, choose **In data points**.

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

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
COMSOL Solution

Point Graph 2

1 Right-click **Results>1D Plot Group 5>Point Graph 1** and choose **Duplicate**.

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

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

4 From the **Eigenfrequency selection** list, choose **First**.

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

6 Locate the **Legends** section. Clear the **Show legends** check box.

Global 1

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 5** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
int1(Vdc)		

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

5 From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `Vdc`.

7 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`Experiment: Bannon et. al.`

1D Plot Group 5

1 In the **Model Builder** window, under **Results** click **1D Plot Group 5**.

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 `Eigenfrequency vs. DC voltage`.

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

6 In the associated text field, type `DC Bias (V)`.

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

8 In the associated text field, type `Resonant Frequency (Hz)`.

9 Right-click **Results>1D Plot Group 5** and choose **Rename**.

10 In the **Rename 1D Plot Group** dialog box, type `Eigenfrequency vs DC Voltage` in the **New label** text field.

11 Click **OK**.

Compare this plot with that in . Note the spring softening effect.

Created in COMSOL Multiphysics 5.3

Pull-In Voltage for a Biased Resonator—3D

Introduction

Silicon micromechanical resonators have long been used for designing sensors and are now becoming increasingly important as oscillators in the consumer electronics market. In this sequence of models, a surface micromachined MEMS resonator, designed as part of a micromechanical filter, is analyzed in detail. The resonator is based on that developed in Ref. 1.

This model performs a pull-in analysis of the structure, to predict the point at which the biased system becomes unstable. The analysis begins from the stationary analysis performed in the accompanying model Stationary Analysis of a Biased Resonator—3D; please review this model first.

Model Definition

The geometry, fabrication, and operation of the device are discussed for the “Stationary Analysis of a Biased Resonator—3D” model.

This model computes the pull-in voltage for the resonator by solving an inverse problem. The z-coordinate of the resonator midpoint is computed using an integration operator (`intop1`). The inverse problem that COMSOL solves computes the DC voltage that must be applied to the beam in order to move the midpoint to a set z-coordinate, `zset`. This is achieved by adding a global equation for the DC voltage, `VdcSP`, applied to the resonator. The equation `intop1(z)-zset=0` is solved to determine the value of `VdcSP`. This means that `VdcSP` is adjusted until the midpoint of the resonator has a z-coordinate given by the set value, `zset`. Essentially COMSOL is being asked to find the voltage that allows the beam to exist in equilibrium (stable or unstable) at a given displacement. Solving the problem in this way avoids complications with trying to solve a problem with no solution (which is what happens if the voltage is continuously ramped up eventually exceeding the pull-in voltage). The result of the analysis is a displacement versus voltage plot, with a minimum at the pull-in voltage. Note that for a linear spring, the pull-in displacement corresponds to 1/3 of the gap distance. Although the inclusion of geometric nonlinearities in the solid mechanics solver means that the pull-in displacement changes slightly from this value, it is usually most efficient to search around this point for the pull-in voltage.

Results and Discussion

Figure 1 shows the voltage-displacement curve for the resonator at equilibrium. The y-coordinate at which the pull in occurs corresponds to a displacements around 1/3 of the gap size. The pull-in voltage is 59.1 V.

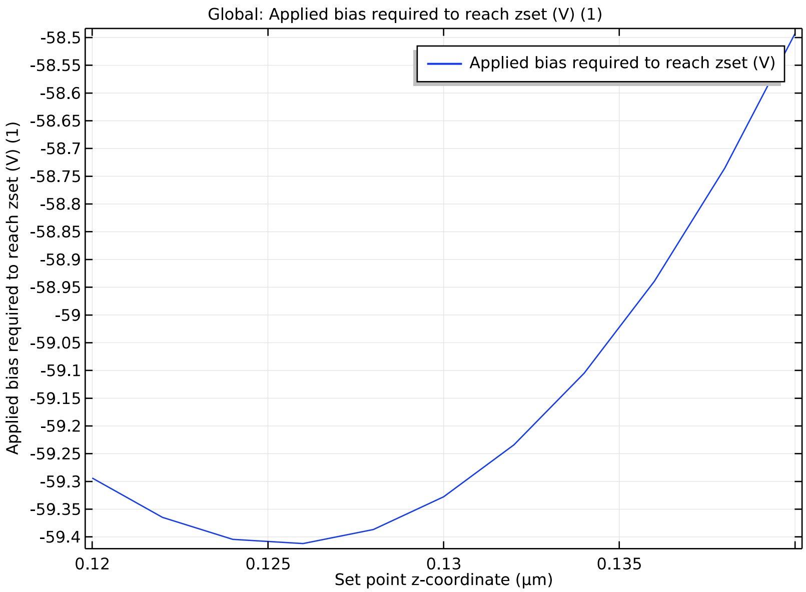


Figure 1: Voltage required to achieve a set displacement versus the target displacement. The pull-in voltage is the minimum of the plot: 59.4 V.

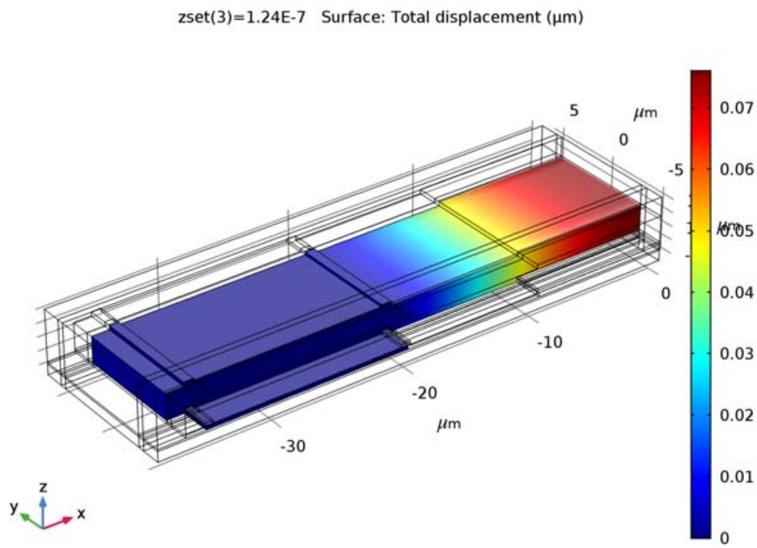


Figure 2: z-displacement of the resonator at pull-in. The displacement at pull-in is 75 nm. For a linear spring the displacement at pull in would be 66 nm.

Figure 2 shows the z-displacement of the resonator at the pull-in voltage. The maximum displacement at pull-in is 75 nm. This is comparable to the linear spring value of 66 nm.

Notes About the COMSOL Implementation

To compute the voltage required to generate the desired displacement of the beam, use a global equation. A common use of global equations is for computing the value of a dependent variable based on an ordinary differential equation in the dependent variable itself. However, it is also possible to couple a global equation with the other PDEs in the model as a powerful tool to solve certain kinds of inverse problems. This model uses a global equation to compute the potential applied to the drive electrode. The equation takes the form

$$z_0 = z_{\mathrm{set}}$$

where z_0 is the z-coordinate of the midpoint of the beam's underside and z_{set} is the desired z-coordinate. COMSOL Multiphysics computes the voltage to satisfy the constraint implied by the above equation. Note that the large difference in scale between the set-point displacement (10^{-7} m) and the applied voltages (10^{2} V) means that care must be taken with the dependent variable scaling in the solver settings.

Reference

1. F.D. Bannon III, J.R. Clark and C.T.-C. Nguyen, “High-Q HF Microelectromechanical Filters,” *IEEE Journal of Solid State Circuits,* vol. 35, no. 4, pp. 512–526, 2000.

Application Library path: `MEMS_Module/Actuators/biased_resonator_3d_pull_in`

Modeling Instructions

Open the existing stationary study (filename: biased_resonator_3d_basic.mph).

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

Browse to the application’s Application Libraries folder and double-click the file `biased_resonator_3d_basic.mph`.

Add a parameter to set the z-coordinate of the midpoint of the resonator.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, expand the **Global Definitions** node, then click **Parameters**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.
3. In the table, enter the following settings:

Name	Expression	Value	Description
zset	100[nm]	1E-7 m	Set point z-coordinate

Add an integration operator to compute the actual displacement.

DEFINITIONS

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 253 only.

Change the drive potential to the value `VdcSP` - which will be solved for in a global equation.

ELECTROMECHANICS (EMI)

Electric Potential 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Electromechanics (emi)** node, then click **Electric Potential 1**.

2 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

3 In the V_0 text field, type `VdcSP`.

Add a global equation to compute the voltage for a given displacement, `VdcSP`.

4 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

Global Equations 1

1 On the **Physics** toolbar, click **Global** and choose **Global Equations**.

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

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
VdcSP	intop1(z)-zset	0	0	Applied bias required to reach zset (V)

Set up a parametric sweep over `zset`.

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**, click to expand the **Study extensions** section.
3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
4 Click **Add**.
5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
zset		

6 Click **Range**.
7 In the **Range** dialog box, type `120e-9` in the **Start** text field.
8 In the **Step** text field, type `2e-9`.
9 In the **Stop** text field, type `140e-9`.
10 Click **Replace**.

The dependent variables require scaling correctly in order to assist the solver.

Solution 2 (sol2)

1 On the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Dependent Variables 1**.
3 In the **Settings** window for **Dependent Variables**, locate the **General** section.
4 From the **Defined by study step** list, choose **User defined**.
5 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** node, then click **Electric potential (comp1.V)**.
6 In the **Settings** window for **Field**, locate the **Scaling** section.
7 From the **Method** list, choose **Manual**.
8 In the **Scale** text field, type `100`.
9 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Spatial coordinates (comp1.xyz)**.
10 In the **Settings** window for **Field**, locate the **Scaling** section.
11 From the **Method** list, choose **Manual**.
12 In the **Scale** text field, type `1e-6`.

13 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Applied bias required to reach zset (V) (comp1.ODE1)**.

14 In the **Settings** window for **State**, locate the **Scaling** section.

15 From the **Method** list, choose **Manual**.

16 In the **Scale** text field, type `100`.

The problem is highly non-linear, so the solver settings need to be adjusted accordingly.

17 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node.

18 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** and choose **Fully Coupled**.

19 In the **Settings** window for **Fully Coupled**, locate the **General** section.

20 From the **Linear solver** list, choose **Direct**.

21 Click to expand the **Method and termination** section. Locate the **Method and Termination** section. From the **Nonlinear method** list, choose **Automatic highly nonlinear (Newton)**.

22 In the **Model Builder** window, right-click **Study 2** and choose **Rename**.

23 In the **Rename Study** dialog box, type `Pull In` in the **New label** text field.

24 Click **OK**.

25 On the **Study** toolbar, click **Compute**.

RESULTS

Displacement (emi) 1

Determine the pull-in voltage by plotting `VdcSP` vs. `zset`.

1D Plot Group 6

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 **Pull In/Solution 2 (sol2)**.

Global 1

1 Right-click **1D Plot Group 6** and choose **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>VdcSP - Applied bias required to reach zset (V)**.

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

4 In the **Expression** text field, type `zset`.

1D Plot Group 6

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 6** and choose **Rename**.
2 In the **Rename 1D Plot Group** dialog box, type `Pull-In Plot` in the **New label** text field.
3 Click **OK**.
4 On the **Pull-In Plot** toolbar, click **Plot**.
5 Click the **Zoom Extents** button on the **Graphics** toolbar.

Compare the resulting plot with . The pull in voltage is the minimum of the curve: 59.4 V at zset=126 nm.

Now look at the displacement at pull in. The default plot group can be used.

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 **Parameter value (zset)** list, choose **1.24E-7**.
4 On the **Displacement (emi) 1** toolbar, click **Plot**.

Compare the resulting plot with . The displacement at pull-in is 75 nm.

Created in COMSOL Multiphysics 5.3

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.

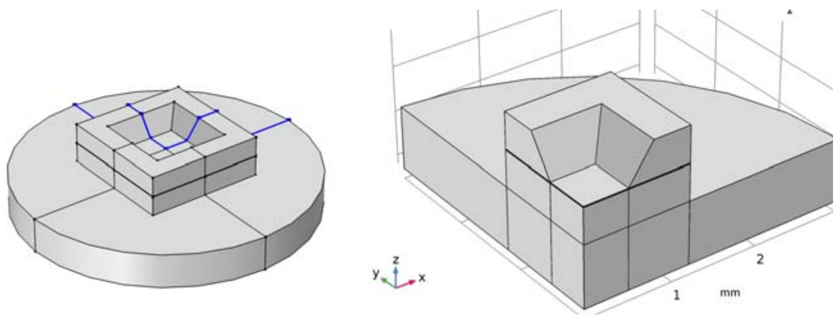


Figure 1: The model geometry. Left: The symmetric device geometry, with one quadrant highlighted in blue, showing the symmetry planes. Right: In COMSOL only the highlighted quadrant is modeled, and the symmetry boundary condition is used on the cross section walls.

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.).

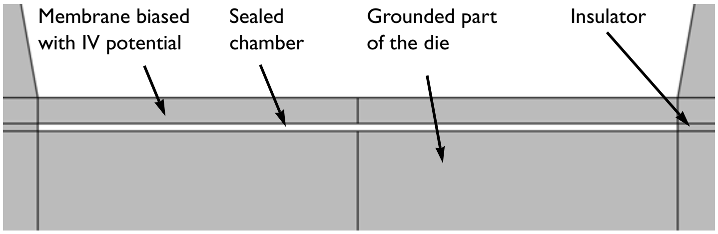


Figure 2: Cross section through the device showing the capacitor. The vertical axis has been expanded to emphasize the gap.

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.

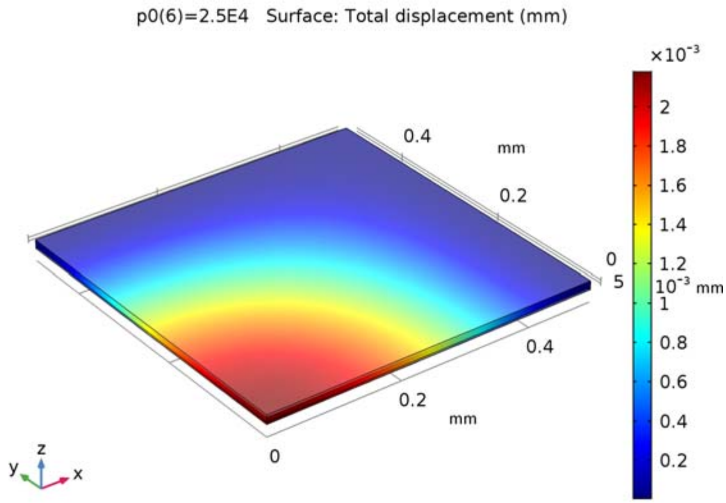


Figure 3: Quadrant deflection when the pressure difference across the membrane is 25 kPa. As expected the deflection is greatest in the center of the membrane

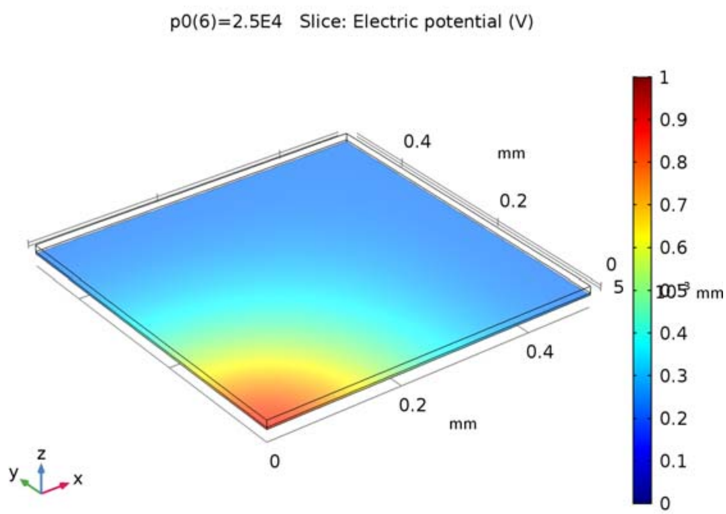


Figure 4: Electric potential in the air chamber, plotted on a slice between the two plates of the capacitor. The potential has become nonuniform as a result of the pressure-induced deformation of the diaphragm.

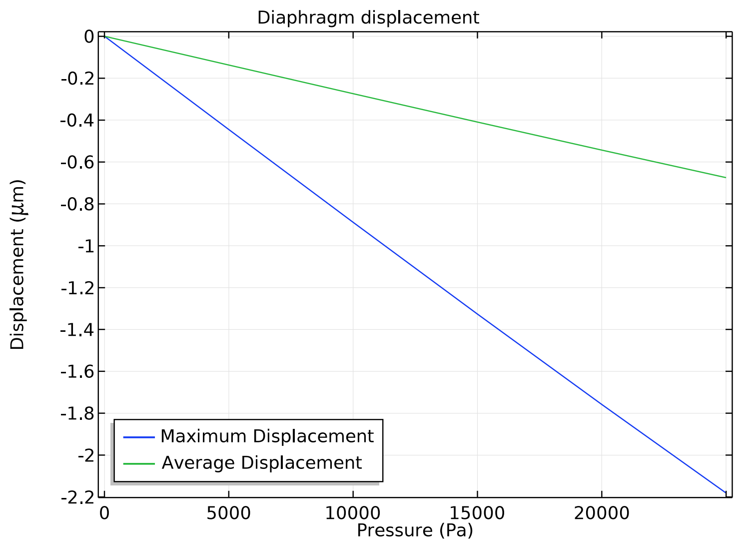


Figure 5: Maximum and mean displacement of the membrane as a function of the applied pressure.

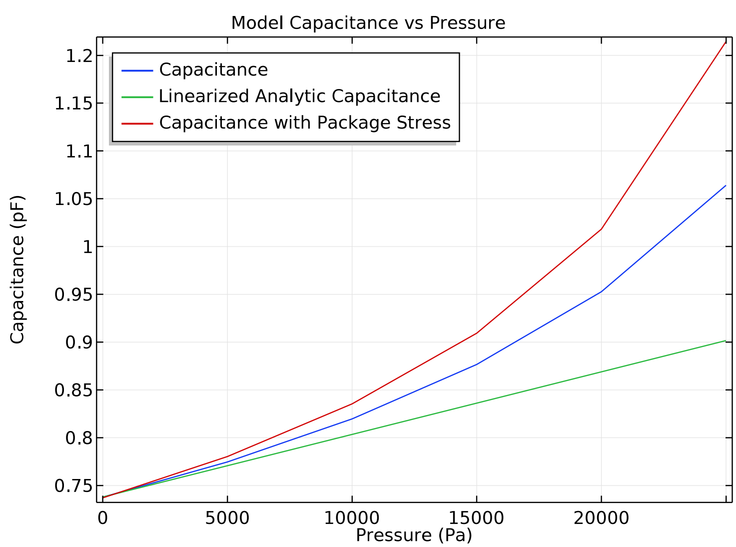


Figure 6: Capacitance of the membrane as a function of applied pressure, both with and without the packaging stresses. The linearized zero pressure capacitance variation, taken from Ref. 1, is also shown for comparison.

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 7.3×10^{-6} pF/Pa (compare to the value of 6.5×10^{-6} pF/Pa given in Ref. 1). The device sensitivity is therefore 29×10^{-6} pF/Pa (compare to 26×10^{-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 μV/Pa (compared to 26 μV/Pa from Ref. 1). Using a smaller pressure step to produce the plot improves the agreement leading to a response at the origin of 6.7×10^{-6} pF/Pa (27×10^{-6} pF/Pa for the device, corresponding to 27 μV/Pa). The response is nonlinear, so that at 20 kPa the model output is 14.3×10^{-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 17×10^{-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.

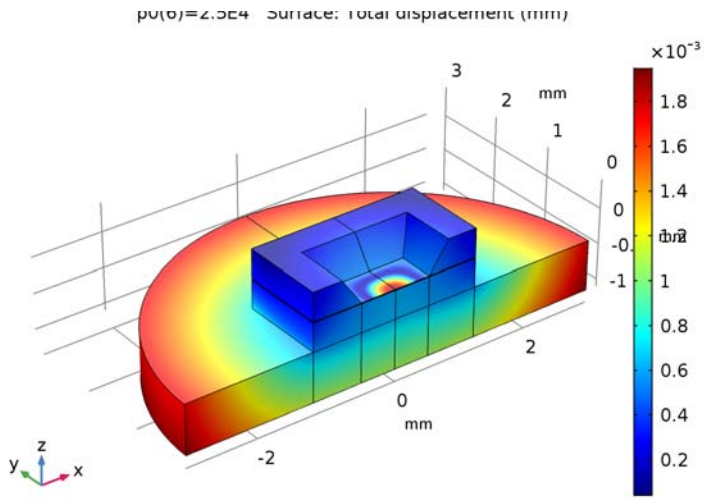


Figure 7: The displacement of the structure due to an applied pressure of 25 kPa when packaging stresses are also included in the model. Displacements are shown at the operating temperature of 20 °C, and are assumed to be zero at the die bonding temperature of 70 °C.

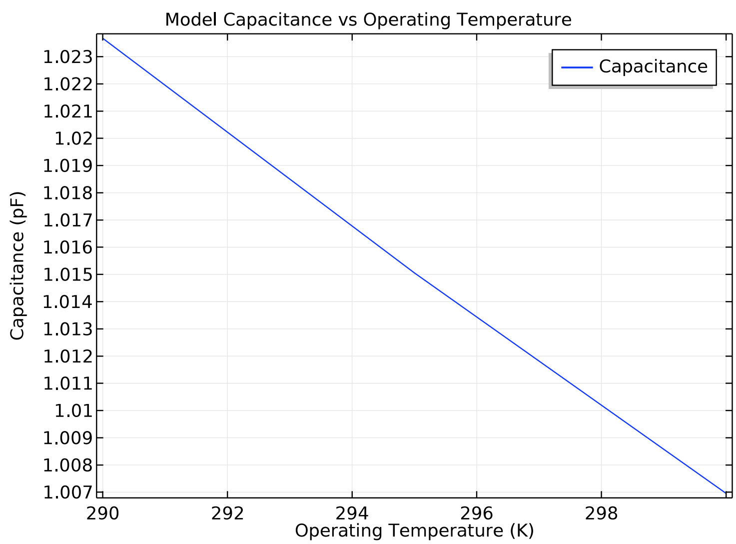


Figure 8: Temperature dependence of the capacitance of the packaged device. The capacitance varies with temperature as a result of temperature induced changes in the packaging stress within the diaphragm.

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 6.5×10^{-6} pF/Pa to 10×10^{-6} pF/Pa (40×10^{-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 25×10^{-6} pF/Pa (100 pF/Pa for the entire device) compared to the unstressed value of 14.3×10^{-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 3.5×10^{-3} pF/K (14×10^{-3} pF/K for the whole device). With a pressure sensitivity of 25×10^{-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 unstressed 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

1. V. Kaajakari, *Practical MEMS,* Small Gear Publishing, Las Vegas, 2009.

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**.

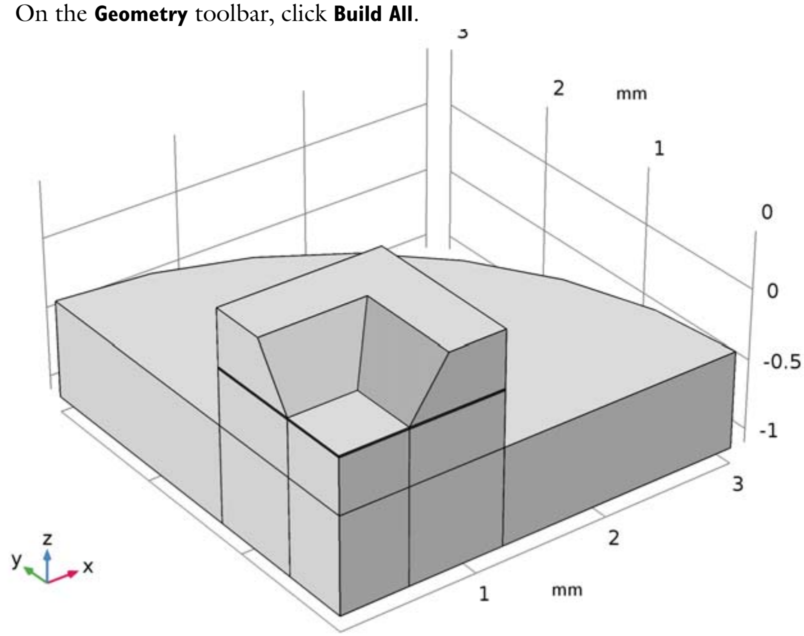
For convenience, the device geometry is inserted from an existing file. You can read the instructions for creating the geometry in the .

GEOMETRY 1

1 On the **Geometry** toolbar, click **Insert Sequence**.

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

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



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:

Name	Expression	Value	Description
`p0`	`20[kPa]`	2E4 Pa	`Pressure`
`T0`	`20[degC]`	293.2 K	`Operating temperature`
`Tref`	`70[degC]`	343.2 K	`Die Bonding temperature`

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 `-100[um]`.

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 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromechanics (emi)** 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 In the **Model Builder** window, right-click **Electromechanics (emi)** and choose the boundary condition **Structural>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 Right-click **Electromechanics (emi)** and choose the boundary condition **Structural>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 Right-click **Electromechanics (emi)** and choose **Points>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 Right-click **Electromechanics (emi)** and choose the boundary condition **Structural>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 In the **Model Builder** window, right-click **Electromechanics (emi)** and choose the boundary condition **Electrical>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 Right-click **Electromechanics (emi)** and choose the boundary condition **Electrical> 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 . 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

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	`11.7`	1	Basic
Young's modulus	E	`170[GPa]`	Pa	Basic
Poisson's ratio	nu	`0.06`	1	Basic
Density	rho	2330	kg/m^3	Basic

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

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

6 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)

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

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

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

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

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	1	1	Basic

5 Click to expand the **Material properties** section. Locate the **Material Properties** section. From the **Material type** list, choose **Nonsolid**.

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

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

8 Click **OK**.

ADD MATERIAL

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

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

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

4 Click **Add to Component** in the window toolbar.

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)>Mesh 1** 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 `50[um]`.

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.

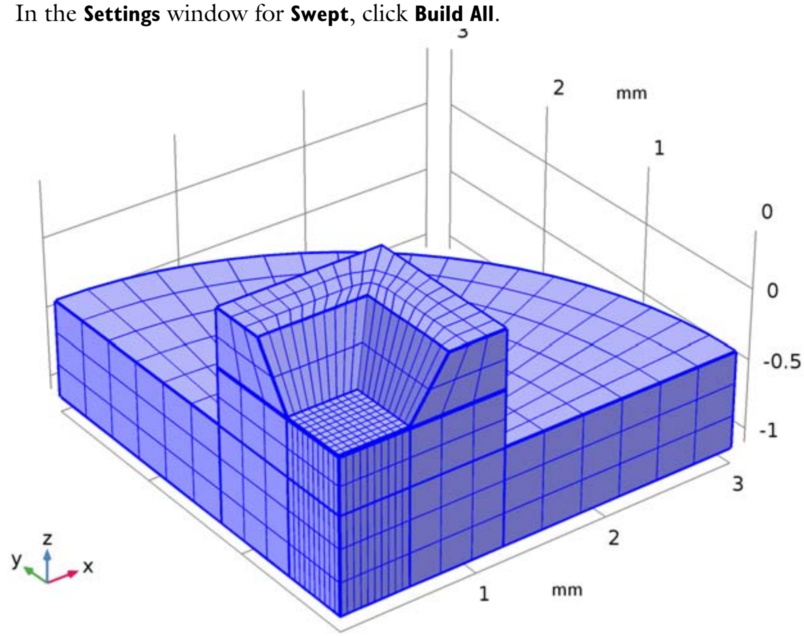
3 In the **Settings** window for **Mapped**, click **Build All**.

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 `p0 (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**.

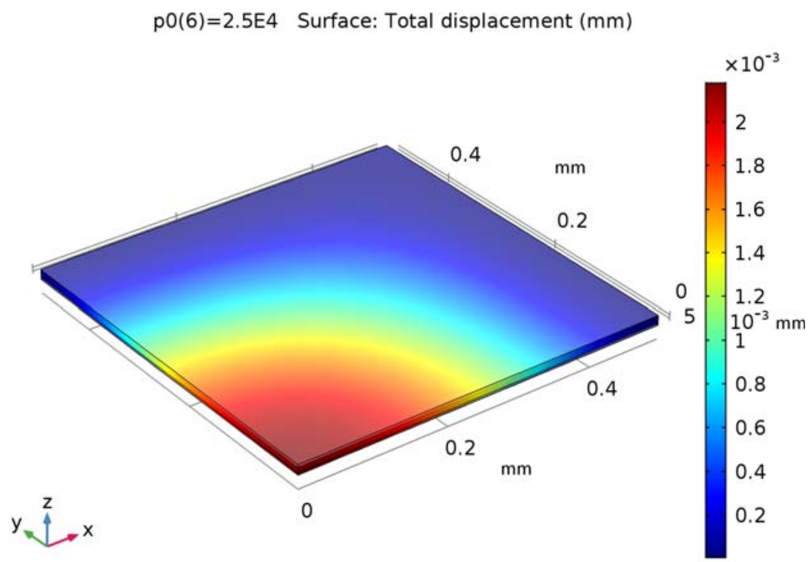
2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

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**.

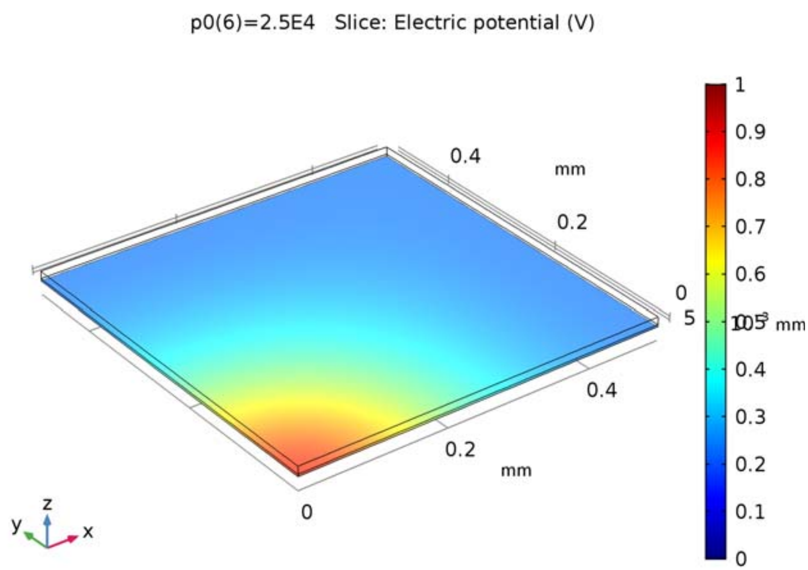
4 In the **Planes** text field, type `1`.

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.

Global 1

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

2 In the **Model Builder** window, right-click **1D Plot Group 3** and choose **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.

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

4 In the table, enter the following settings:

Expression	**Unit**	**Description**
`intop1(w)`	`um`	`Maximum Displacement`
`aveop1(w)`	`um`	`Average Displacement`

1D Plot Group 3

1 In the **Model Builder** window, under **Results** click **1D 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)`.

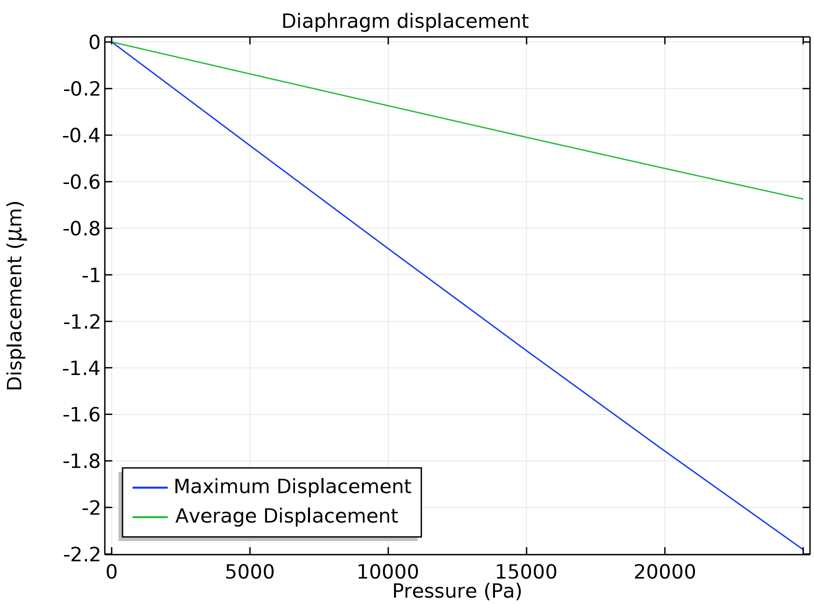
9 Click to expand the **Legend** section. From the **Position** list, choose **Lower left**.

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 um. The average displacement of the diaphragm is 0.27 υm. These values are in good agreement with the approximate model given in (maximum displacement 0.93 υm, average displacement 0.27 υm).

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

Global 1

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

2 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **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.

3 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 - Maxwell capacitance**.

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

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

Expression	Unit	Description
`emi.C11`	`pF`	`Capacitance`
`0.738[pF]*(1+8.87e-6[1/Pa]*p0)`	`pF`	`Linearized Analytic Capacitance`

1D Plot Group 4

1 In the **Model Builder** window, under **Results** click **1D 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)`.

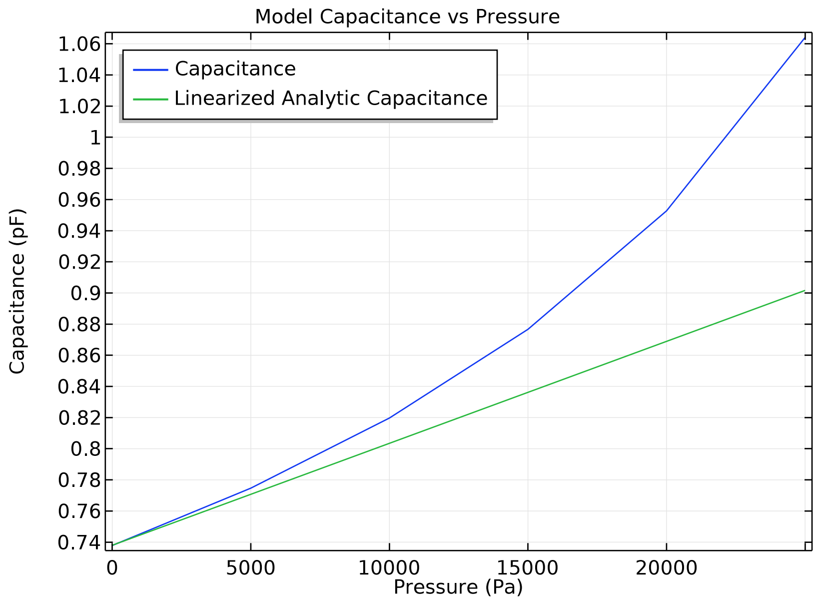
9 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

10 Right-click **Results>1D 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 this corresponds to a sensor transfer function of 29 υV/Pa for the COMSOL model and 26 υV/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)

Thermal Expansion 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromechanics (emi)** right-click **Linear Elastic Material 1** 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 `T0`.

   The reference temperature indicates the temperature at which the structure had no thermal strains. In this case, set it to the previously defined parameter, `Tref`, 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_{ref} text field, type `Tref`.

   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 **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silicon (mat1)**.

2 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 `p0 (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

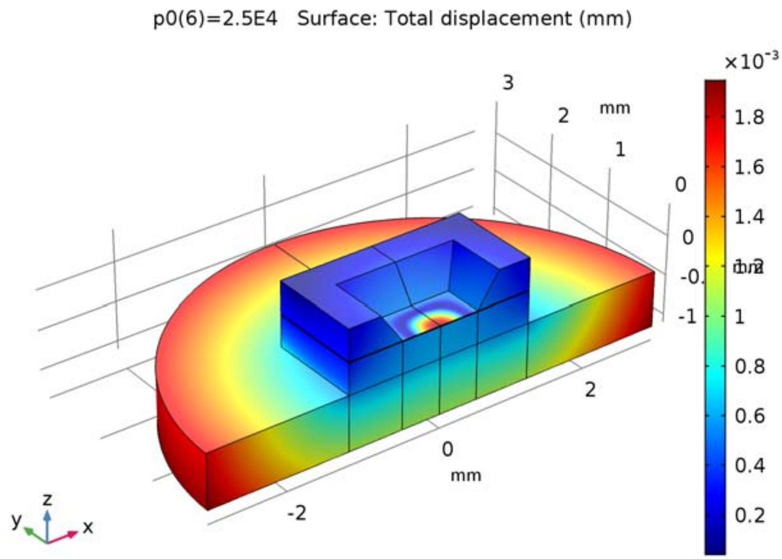
1 In the **Model Builder** window, under **Results** click **Displacement (emi) 1**.

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) 1** 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 2

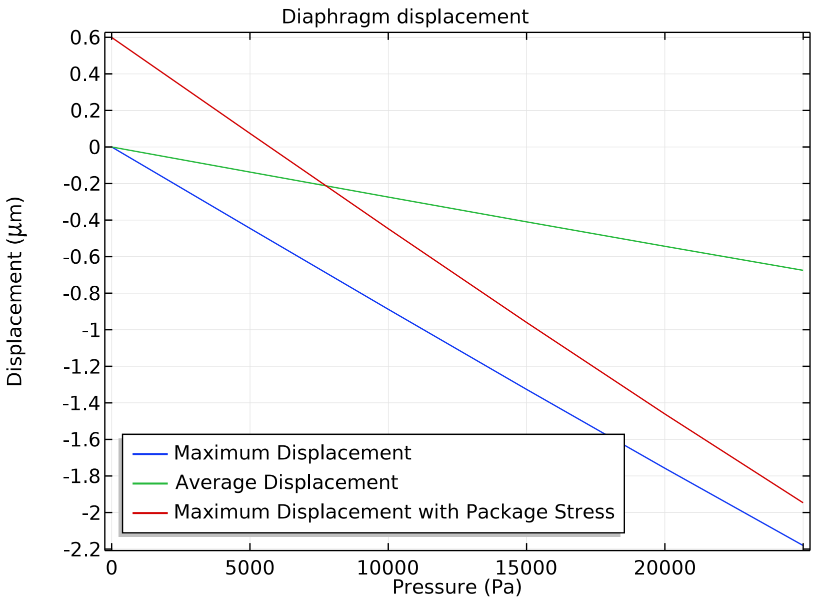
1 In the **Model Builder** window, under **Results>Diaphragm Displacement vs Pressure** right-click **Global 1** and choose **Duplicate**.

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

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

Note that the `aveop1(w)` expression has been removed from the table.

4 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.

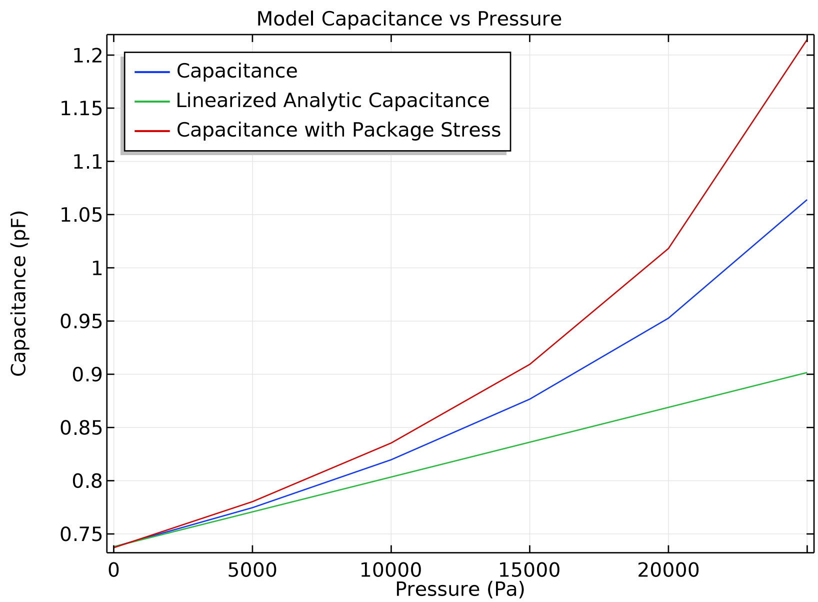
Global 2

1. In the **Model Builder** window, under **Results** right-click **Model Capacitance vs Pressure** and choose **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 - Maxwell capacitance**.
5. Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`emi.C11`	`pF`	`Capacitance with Package Stress`

6 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).



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:

Parameter name	Parameter value list	Parameter unit
T0		

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.

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

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 p0, 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 Right-click **1D Plot Group 7** and choose **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 - Maxwell capacitance**.

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

Expression	**Unit**	**Description**
`emi.C11`	`pF`	`Capacitance`

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.

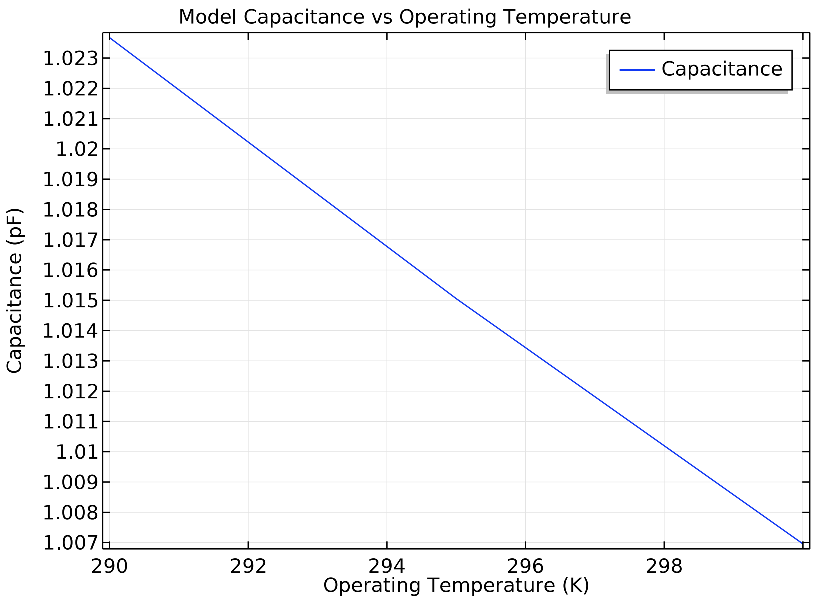
6 In the associated text field, type `Operating Temperature (K)`.

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 (0.6Pa) this number is very large. This model shows that a naive choice of packaging can have a highly detrimental effect on sensor performance.

Appendix — Geometry Instructions

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

NEW

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

ROOT

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

GEOMETRY 1

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 **mm**.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

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

3 In the **Width** text field, type `1.2`.

4 In the **Depth** text field, type `1.2`.

5 In the **Height** text field, type `1.51`.

6 Locate the **Position** section. In the **z** text field, type `-1.1`.

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

Layer name	Thickness (mm)
`Layer 1`	`0.7`
`Layer 2`	`0.397`
`Layer 3`	`0.003`
`Layer 4`	`0.01`

Block 2 (blk2)

1 On the **Geometry** toolbar, click **Block**.

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

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

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

5 In the **Height** text field, type `1.51`.

6 Locate the **Position** section. In the **z** text field, type `-1.1`.

Partition Domains 1 (pard1)

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

2 On the object **blk1**, select Domains 1–4 only.

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

4 From the **Partition with** list, choose **Objects**.

5 Select the object **blk2** only.

Hexahedron 1 (hex1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Hexahedron**.

2 In the **Settings** window for **Hexahedron**, locate the **Vertices** section.

3 In row **2**, set **x** to 0.5.

4 In row **3**, set **x** to 0.5.

5 In row **4**, set **x** to 0.

6 In row **6**, set **x** to 0.78322.

7 In row **7**, set **x** to 0.78322.

8 In row **8**, set **x** to 0.

9 In row **2**, set **y** to 0.

10 In row **3**, set **y** to 0.5.

11 In row **4**, set **y** to 0.5.

12 In row **6**, set **y** to 0.

13 In row **7**, set **y** to 0.78322.

14 In row **8**, set **y** to 0.78322.

15 In row **1**, set **z** to 0.01.

16 In row **2**, set **z** to 0.01.

17 In row **3**, set **z** to 0.01.

18 In row **4**, set **z** to 0.01.

19 In row **5**, set **z** to 0.41.

20 In row **6**, set **z** to 0.41.

21 In row **7**, set **z** to 0.41.

22 In row **8**, set **z** to 0.41.

Difference 1 (dif1)

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

2 Select the object **pard1** only.

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

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **hex1** only.

Cylinder 1 (cyl1)

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

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

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

4 In the **Height** text field, type 0.7.

5 Locate the **Position** section. In the **z** text field, type `-1.1`.

Partition Domains 2 (pard2)

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

2 On the object **cyl1**, select Domain 1 only.

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

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

5 On the object **dif1**, select Boundaries 17 and 38 only.

6 In the tree, select **dif1**.

7 Click **Build Selected**.

8 On the **Geometry** toolbar, click **Delete**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Delete Entities 1 (del1)**.

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

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

4 On the object **pard2**, select Domains 1–3 only.

5 On the object **blk2**, select Domain 1 only.

Form Union (fin)

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

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Explicit Selection 1 (sel1)

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

2 In the **Settings** window for **Explicit Selection**, type `YZ Symmetry Plane` 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** check box.

5 On the object **fin**, select Boundaries 1, 4, 7, 10, 14, 17, 20, 23, 26, and 30 only.

Explicit Selection 2 (sel2)

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

2 In the **Settings** window for **Explicit Selection**, type `XZ Symmetry Plane` 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** check box.

5 On the object **fin**, select Boundaries 2, 5, 8, 11, 40, 42, 44, 46, 48, and 50 only.

Box Selection 1 (boxsel1)

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

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

3 Locate the **Box Limits** section. In the **z maximum** text field, type `-0.1`.

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

Explicit Selection 3 (sel3)

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

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

3 On the object **fin**, select Domain 3 only.

Explicit Selection 4 (sel4)

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

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

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

4 Select the object **fin** only.

Difference Selection 1 (difsel1)

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

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

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

4 In the **Add** dialog box, select **Geometry** in the **Selections to add** list.

5 Click **OK**.

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

7 Click **Add**.

8 In the **Add** dialog box, select **Cavity** in the **Selections to subtract** list.

9 Click **OK**.

Created in COMSOL Multiphysics 5.3

Composite Piezoelectric Transducer

Introduction

This example shows how to set up a piezoelectric transducer model following the work of Y. Kagawa and T. Yamabuchi (Ref. 1). The composite piezoelectric ultrasonic transducer has a cylindrical geometry that consists of a piezoceramic (NEPEC 6) layer, two aluminum layers, and two adhesive layers. The layers are organized as follows: aluminum layer–adhesive layer–piezoceramic layer–adhesive layer–aluminum layer.

The system applies an AC potential on the electrode surfaces of both sides of the piezoceramic layer. The potential in this example has a peak value of 1 V in the frequency range 20 kHz to 106 kHz. The goal is to compute the susceptance (the imaginary part of the admittance) $Y = I/V$, where I is the total current and V is the potential, for a frequency range around the four lowest eigenfrequencies of the structure.

The first step finds the eigenmodes, and the second step runs a frequency sweep across an interval that encompasses the first four eigenfrequencies. Both analyses are fully coupled, and COMSOL Multiphysics assembles and solves both the electric and mechanical parts of the problem simultaneously.

Although you could analyze this problem using a 2D axisymmetric model, in order to illustrate the modeling principles for more complicated problems, this example uses a 3D geometry.

When creating the model geometry, you make use of the symmetry by first making a cut along a midplane perpendicular to the central axis and then by cutting out a 10-degree wedge; doing so reduces memory requirements significantly.

Model Data

The model uses the following material data.

NEPEC 6 MATERIAL PARAMETERS

TABLE 1: ELASTICITY MATRIX c_E

128 GPa	68 GPa	66 GPa	0	0	0
	128 GPa	66 GPa	0	0	0
		110 GPa	0	0	0
			21 GPa	0	0
				21 GPa	0
					21 GPa

TABLE 2: COUPLING MATRIX e

0	0	0	0	0	0
0	0	0	0	0	0
-6.1	-6.1	15.7	0	0	0

TABLE 3: RELATIVE PERMITTIVITY ε_{rS}

993.53	0	0
	993.53	0
		993.53

ALUMINUM MATERIAL PARAMETERS

PARAMETER	EXPRESSION/VALUE	DESCRIPTION
E	70.3 GPa	Young's modulus
nu	0.345	Poisson's ratio
rho	2690	Density

ADHESIVE MATERIAL PARAMETERS

PARAMETER	EXPRESSION/VALUE	DESCRIPTION
E	10 GPa	Young's modulus
nu	0.38	Poisson's ratio
rho	1700	Density

Results and Discussion

Figure 1 shows the lowest vibration mode of the piezoelectric transducer, while Figure 2 shows the transducer's input susceptance as a function of the excitation frequency.

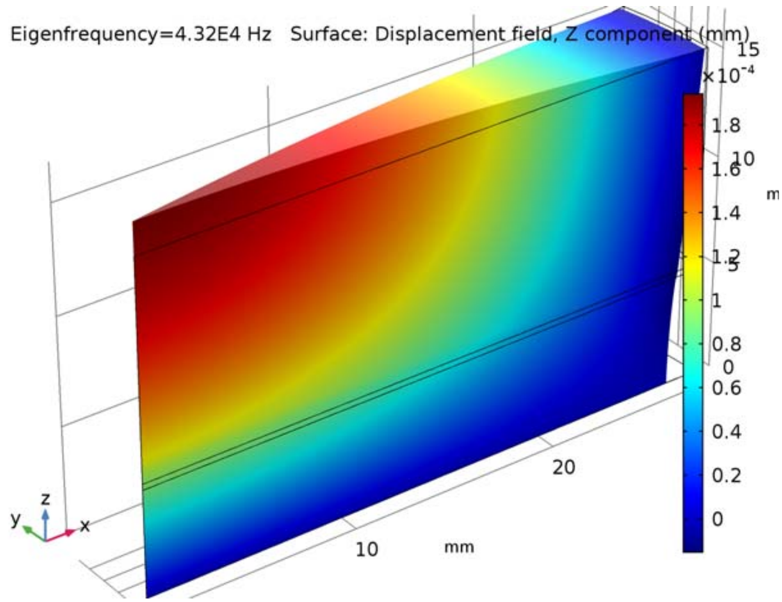


Figure 1: The lowest vibration eigenmode of the transducer.

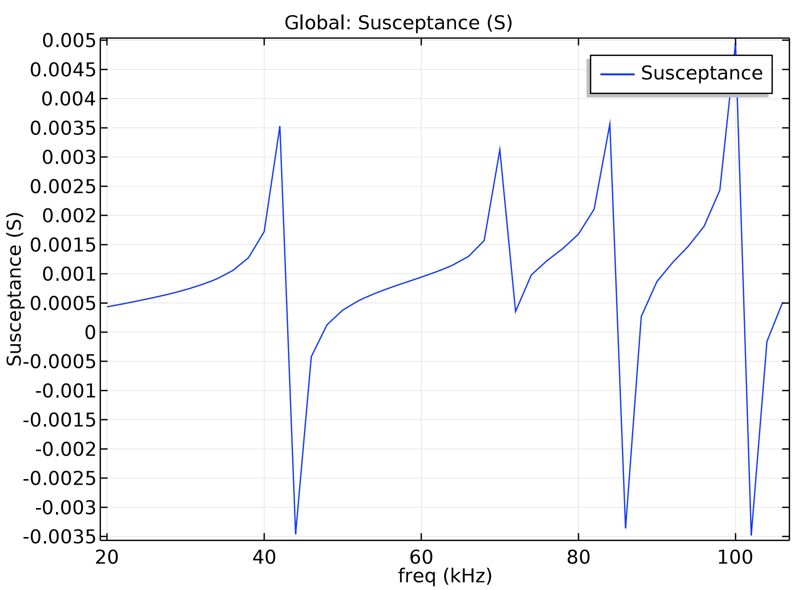


Figure 2: Input susceptance as a function of excitation frequency.

The result is in agreement with the work in Ref. 1. A small discrepancy close to the eigenfrequencies appears because the simulation uses no damping.

Reference

1. Y. Kagawa and T. Yamabuchi, “Finite Element Simulation of a Composite Piezoelectric Ultrasonic Transducer,” *IEEE Transactions on Sonics and Ultrasonics*, vol. SU-26, no. 2, pp. 81–88, 1979.

Application Library path: `MEMS_Module/Piezoelectric_Devices/composite_transducer`

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>Piezoelectric Devices**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Eigenfrequency**.

6 Click **Done**.

GEOMETRY 1

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 **mm**.

Plane Geometry

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

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

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

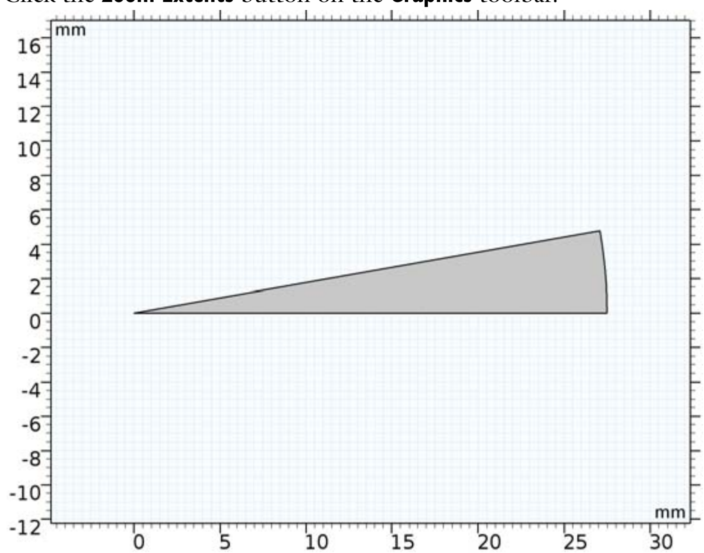
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `27.5`.

4 In the **Sector angle** text field, type `10`.

5 Right-click **Circle 1 (c1)** and choose **Build Selected**.

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



7 In the **Model Builder** window, click **Geometry 1**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

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

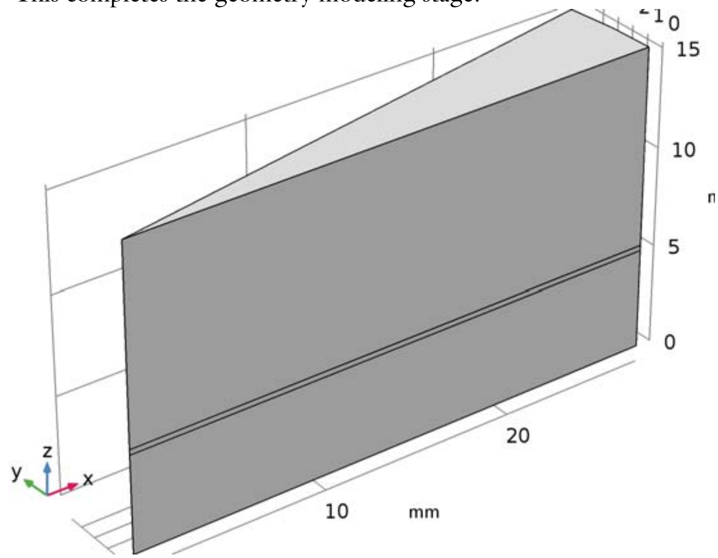
3 In the table, enter the following settings:

Distances (mm)
5
5.275
15.275

4 Click **Build All Objects**.

5 Click **Go to Default View**.

This completes the geometry modeling stage.



Before defining material properties, select the domains where each physics applies. Proceeding in this order enables to preselect required material properties during their definition.

SOLID MECHANICS (SOLID)

Piezoelectric Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Piezoelectric Material 1**.

2 In the **Settings** window for **Piezoelectric Material**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.

ELECTROSTATICS (ES)

On the **Physics** toolbar, click **Solid Mechanics (solid)** and choose **Electrostatics (es)**.

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

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

3 Click **Clear Selection**.

4 Select Domain 1 only.

Now materials can be defined.

MATERIALS

Material 1 (mat1)

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

2 In the **Settings** window for **Material**, type `Nepec 6` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

4 Select Domain 1 only.

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

Property	Name	Value	Unit	Property group
Elasticity matrix (Ordering: xx, yy, zz, yz, xz, xy)	{cE11, cE12, cE22, cE13, cE23, cE33, cE14, cE24, cE34, cE44, cE15, cE25, cE35, cE45, cE55, cE16, cE26, cE36, cE46, cE56, cE66} ; cEij = cEji	`{128[GPa], 68[GPa], 128[GPa], 66[GPa], 66[GPa], 110[GPa], 0, 0, 0, 21[GPa], 0, 0, 0, 0, 21[GPa], 0, 0, 0, 0, 0, 21[GPa]}`	Pa	Stress-charge form
Coupling matrix	{eES11, eES21, eES31, eES12, eES22, eES32, eES13, eES23, eES33, eES14, eES24, eES34, eES15, eES25, eES35, eES16, eES26, eES36}	`{0, 0, -6.1, 0, 0, -6.1, 0, 0, 15.7, 0, 0, 0, 0, 0, 0, 0, 0, 0}`	C/m²	Stress-charge form
Relative permittivity	epsilonrS ; epsilonrSii = epsilonrS, epsilonrSij = 0	`993.53`	1	Stress-charge form
Density	rho	`7730`	kg/m³	Basic

Alternatively, to define the symmetric elasticity matrix, `cE`, and the full coupling matrix, `eES`, you can click the **Edit** button below the Output properties table under `Component1>Materials>Nepec 6>Stress-Charge` form in the Model builder and use the matrix input dialogs to enter the data as given in section .

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, type `Adhesive` in the **Label** text field.
3 Select Domain 2 only.
4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Young's modulus	E	`10[GPa]`	Pa	Basic
Poisson's ratio	nu	`0.38`	1	Basic
Density	rho	`1700`	kg/m³	Basic

Material 3 (mat3)

1 Right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, type `Aluminum` in the **Label** text field.
3 Select Domain 3 only.
4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Young's modulus	E	`70.3[GPa]`	Pa	Basic
Poisson's ratio	nu	`0.345`	1	Basic
Density	rho	`2690`	kg/m³	Basic

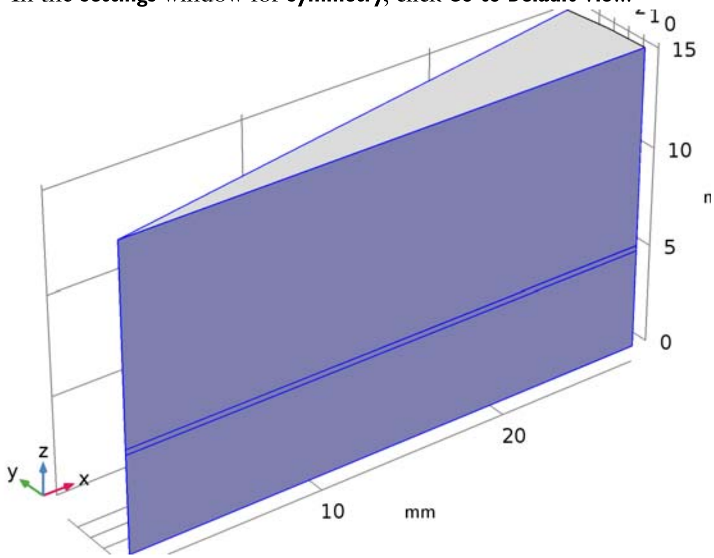
SOLID MECHANICS (SOLID)

Now apply the boundary conditions for each physics.

Symmetry 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **More Constraints>Symmetry**.
2 Select Boundaries 1–5, 7, and 8 only.

3 In the **Settings** window for **Symmetry**, click **Go to Default View**.



ELECTROSTATICS (ES)

Terminal 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electrostatics (es)** and choose the boundary condition **Terminal**.

2 Select Boundary 6 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 `0.5`.

This is half of the total peak voltage between the terminals, which accounts for modeling only the upper half of the transducer.

Ground 1

1 In the **Model Builder** window, right-click **Electrostatics (es)** and choose **Ground**.

2 Select Boundary 3 only.

DEFINITIONS

Before generating the mesh, define a variable for the susceptance.

Variables 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.
2. In the **Settings** window for **Variables**, locate the **Variables** section.
3. In the table, enter the following settings:

Name	Expression	Unit	Description
B	imag(es.Y11)*36/2		Susceptance

In the above expression, the factor 36 compensates for the fact that the total current at the **Terminal** is only computed for a 10 degree wedge of the full transducer. Moreover, the factor 1/2 accounts for the fact that only the upper half of the transducer is modeled because of symmetry in the z direction and hence only half of the actual voltage is applied. Since no damping is modeled, the real part of the admittance `es.Y11` will be zero. This is why it is suitable to evaluate only the imaginary part of the admittance, i.e. the susceptance.

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 **Element size** list, choose **Finer**.

Free Triangular 1

1. Right-click **Component 1 (comp1)>Mesh 1** and choose **More Operations>Free Triangular**.
2. Select Boundary 3 only.

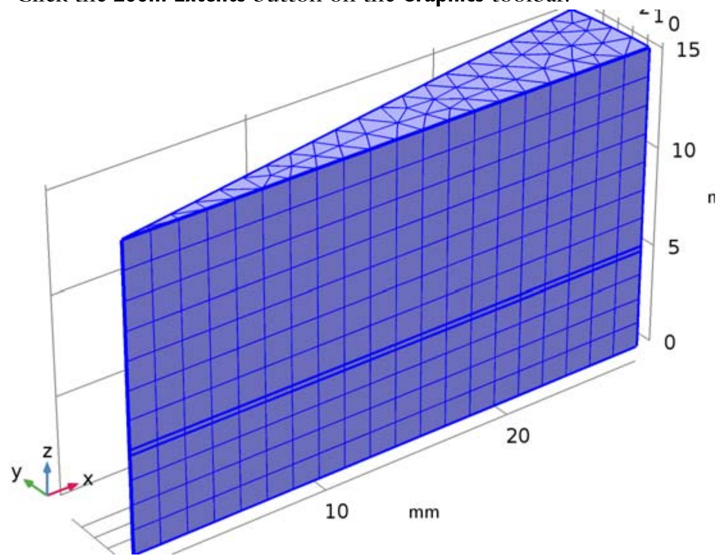
Distribution 1

1. Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Distribution**.
2. Select Edges 2 and 3 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. From the **Distribution properties** list, choose **Predefined distribution type**.
5. In the **Number of elements** text field, type `20`.
6. Click **Build Selected**.

Swept 1

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

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



STUDY 1

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

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Mode Shape (solid)** node, then click **Surface 1**.

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>Solid Mechanics> Displacement>Displacement field (material and geometry frames)>w - Displacement field, Z component**.

3 On the **Mode Shape (solid)** toolbar, click **Plot**.

Compare the resulting plot to that in .

Multislice 1

In the **Model Builder** window, expand the **Electric Potential (es)** node.

Surface 1

1 Right-click **Multislice 1** and choose **Delete**.

2 In the **Model Builder** window, under **Results** right-click **Electric Potential (es)** and choose **Surface**.

3 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electrostatics>Electric>V - Electric potential**.

4 On the **Electric Potential (es)** toolbar, click **Plot**.

Next, add a separate study for the frequency sweep.

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> Frequency Domain**.

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: Frequency Domain

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

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog box, type `20[kHz]` in the **Start** text field.

5 In the **Stop** text field, type `106[kHz]`.

6 In the **Step** text field, type `2[kHz]`.

7 Click **Replace**.

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

RESULTS

Multislice 1

In the **Model Builder** window, expand the **Electric Potential (es) 1** node.

Electric Potential (es) 1

1 Right-click **Multislice 1** and choose **Delete**.

2 On the **Electric Potential (es) 1** toolbar, click **Surface**.

Surface 1

1 In the **Model Builder** window, under **Results>Electric Potential (es) 1** click **Surface 1**.

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>Electrostatics>Electric>V - Electric potential**.

3 On the **Electric Potential (es) 1** toolbar, click **Plot**.

3D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **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 **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 From the **Parameter value (freq (Hz))** list, choose **1.06E5**.

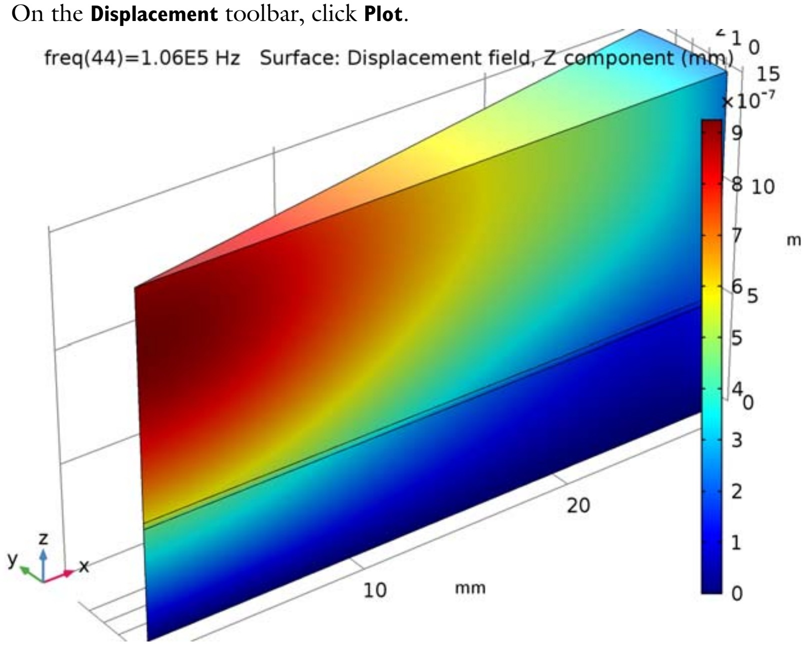
5 On the **Displacement** toolbar, click **Surface**.

Surface 1

1 In the **Model Builder** window, under **Results>Displacement** click **Surface 1**.

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>Solid Mechanics> Displacement>Displacement field (material and geometry frames)>w - Displacement field, Z component**.

3 On the **Displacement** toolbar, click **Plot**.



1D Plot Group 6

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

2 In the **Settings** window for **1D Plot Group**, type `Susceptance` in the **Label** text field.

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

Global 1

1 Right-click **Susceptance** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`B`	`S`	`Susceptance`

4 Locate the **x-Axis Data** section. From the **Unit** list, choose **kHz**.

5 On the **Susceptance** toolbar, click **Plot**.

Compare the result to that in .

Electrostatically Actuated Cantilever

Introduction

The elastic cantilever beam is an elementary structure in MEMS design. This example shows the bending of a beam due to electrostatic forces. The model uses the electromechanics interface to solve the coupled equations for the structural deformation and the electric field. Such structures are frequently tested by means of a low frequency capacitance voltage sweep. The model predicts the results of such a test.

Model Definition

Figure 1 shows the model geometry. The beam has the following dimensions:

- Length: 300 μm
- Width: 20 μm
- Thickness 2 μm

Because the geometry is symmetric only half of the beam needs to modeled. The beam is made of polysilicon with a Young's modulus, E, of 153 GPa, and a Poisson's ratio, ν, of 0.23. It is fixed at one end but is otherwise free to move. The polysilicon is assumed to be heavily doped, so that electric field penetration into the structure can be neglected. The beam resides in an air-filled chamber that is electrically insulated. The lower side of the chamber has a grounded electrode.

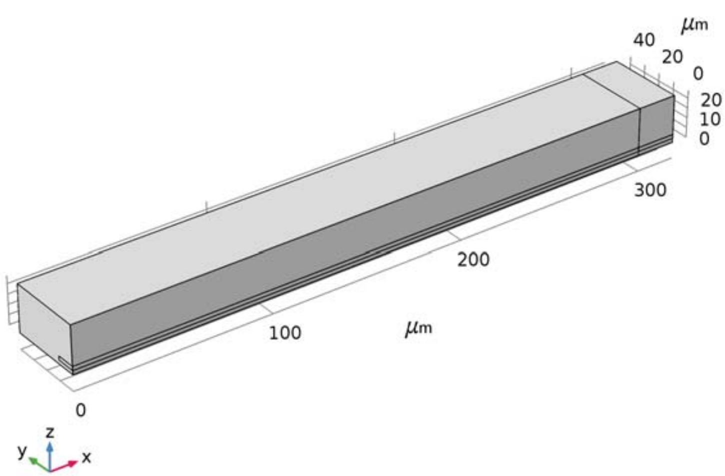


Figure 1: Model Geometry. The beam is 300 μm long and 2 μm thick, and it is fixed at x = 0. The model uses symmetry on the zx-plane at y = 0. The lower boundary of the surrounding air

domain represents the grounded substrate. The model has 20 μm of free air above and to the sides of the beam, while the gap below the beam is 2 μm.

An electrostatic force caused by an applied potential difference between the two electrodes bends the beam toward the grounded plane beneath it. To compute the electrostatic force, this example calculates the electric field in the surrounding air. The model considers a layer of air 20 μm thick both above and to the sides of the beam, and the air gap between the bottom of the beam and the grounded layer is initially 2 μm. As the beam bends, the geometry of the air gap changes continuously, resulting in a change in the electric field between the electrodes. The coupled physics is handled automatically by the Electromechanics interface.

The electrostatic field in the air and in the beam is governed by Poisson's equation:

$$-\nabla \cdot (\varepsilon \nabla V) = 0$$

where derivatives are taken with respect to the spatial coordinates. The numerical model represents the electric potential and its derivatives on a mesh which is moving with respect to the spatial frame. The necessary transformations are taken care of by the Electromechanics interface, which also contains smoothing equations governing the movement of the mesh in the air domain.

The cantilever connects to a voltage terminal with a specified bias potential, V_{in}. The bottom of the chamber is grounded, while all other boundaries are electrically insulated. The terminal boundary condition automatically computes the capacitance of the system.

The force density that acts on the electrode of the beam results from Maxwell's stress tensor:

$$\mathbf{F}_{\mathrm{es}} = -\frac{1}{2}(\mathbf{E} \cdot \mathbf{D})\mathbf{n} + (\mathbf{n} \cdot \mathbf{E})\mathbf{D}$$

where $\mathbf{E}$ and $\mathbf{D}$ are the electric field and electric displacement vectors, respectively, and $\mathbf{n}$ is the outward normal vector of the boundary. This force is always oriented along the normal of the boundary.

Navier's equations, which govern the deformation of a solid, are more conveniently written in a coordinate system that follows and deforms with the material. In this case, these reference or material coordinates are identical to the actual mesh coordinates.

There is positive feedback between the electrostatic forces and the deformation of the cantilever beam. The forces bend the beam and thereby reduce the gap to the grounded substrate. This action, in turn, increases the forces. At a certain voltage the electrostatic forces overcome the stress forces, the system becomes unstable, and the gap collapses. This critical voltage is called the *pull-in voltage*.

At applied voltages lower than the pull-in voltage, the beam stays in an equilibrium position where the stress forces balance the electrostatic forces. Figure 2 shows the beam displacement and the corresponding displacement of the mesh surrounding it. Figure 3 shows the electric potential and electric field that generates these displacements. In Figure 4 the shape of the cantilever's deflection is illustrated for each applied voltage, by plotting the z-displacement of the underside of the beam at the symmetry boundary. The tip deflection as a function of applied voltage is shown in Figure 5. Note that for applied voltages higher than the pull-in voltage, the solution does not converge because no stable stationary solution exists. This situation occurs if an applied voltage of 6.2 V is tried. The pull-in voltage is therefore between 6.1 V and 6.2 V. For comparison, computations in Ref. 1 predict a pull-in voltage of

$$V_{\mathrm{PI}} = \sqrt{\frac{4c_1B}{\varepsilon_0 L^4 c_2^2\left(1 + c_3\dfrac{g_0}{W}\right)}}$$

where $c_1 = 0.07$, $c_2 = 1.00$, and $c_3 = 0.42$; g_0 is the initial gap between the beam and the ground plane; and

$$B = \hat{E}H^3g_0^3$$

If the beam has a narrow width (W) relative to its thickness (H) and length (L), $\hat{E}$ is Young's modulus, E. Otherwise, E and $\hat{E}$, the plate modulus, are related by

$$\frac{E}{\hat{E}} \approx 1 - \nu^2\left(\frac{(W/L)^{1,37}}{0,5 + (W/L)^{1,37}}\right)^{0,98(L/H)^{-0,056}}$$

where ν is Poisson's ratio. Because the calculation in Ref. 1 uses a parallel-plate approximation for calculating the electrostatic force and because it corrects for fringing fields, these results are not directly comparable with those from the simulation. However the agreement is still reasonable: setting $W = 20$ µm results in $V_{\mathrm{PI}} = 6.07$ V.

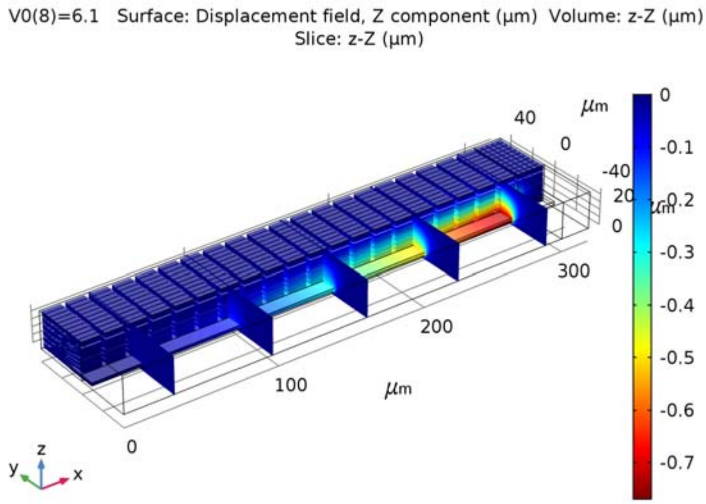


Figure 2: z-displacement for the beam and the moving mesh as a function of position. Each mesh element is depicted as a separate block in the back half of the geometry.

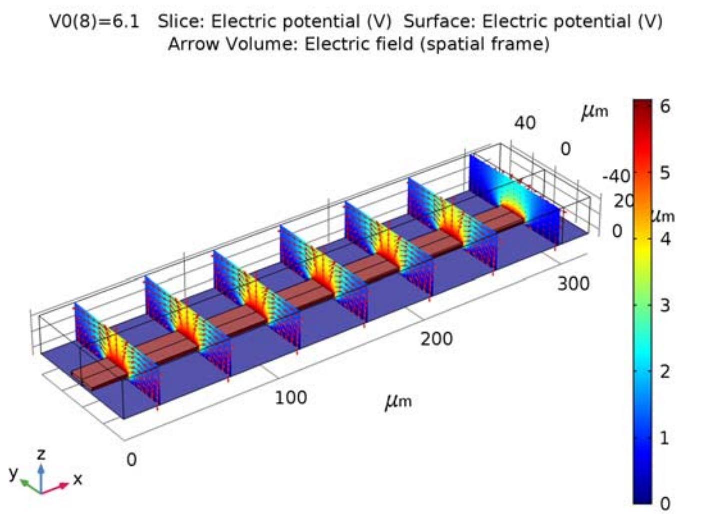


Figure 3: Electric Potential (color) and Electric Field (arrows) at various cross sections through the beam.

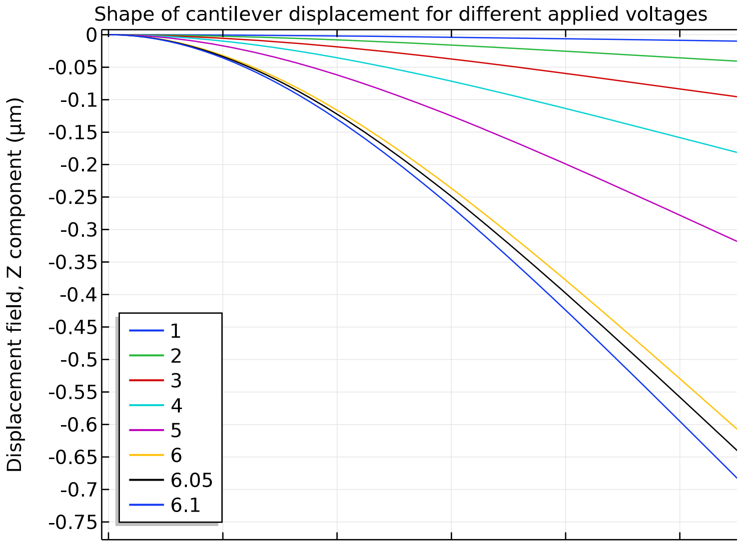


Figure 4: Displacement of the lower surface of the cantilever, plotted along the symmetry boundary, for different values of the applied voltage.

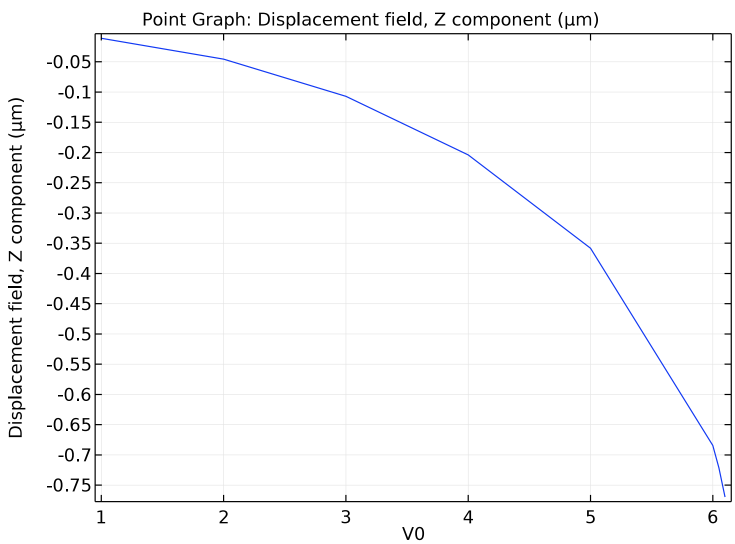


Figure 5: Cantilever tip displacements as a function of applied Voltage V_0.

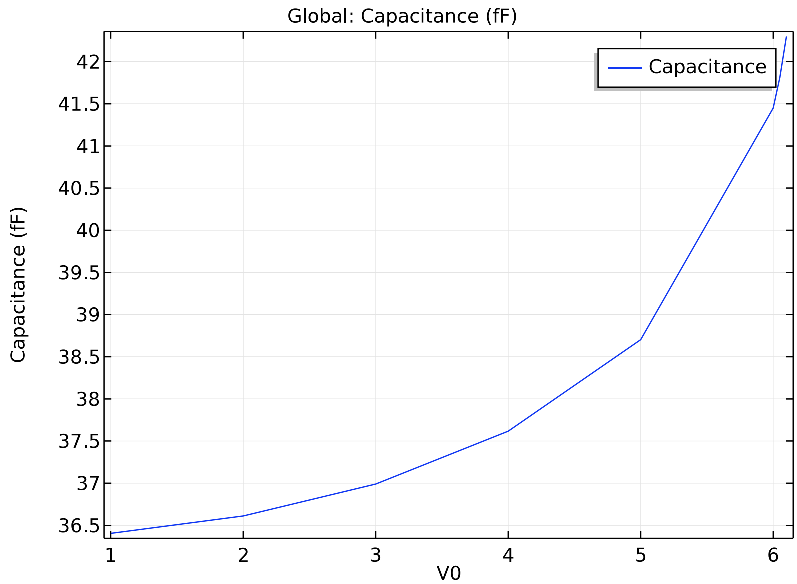


Figure 6: Device capacitance vs applied voltage V_0.

Figure 6 shows the DC C-V curve predicted for the cantilever beam. To some extent, this is consistent with the behavior of an ideal parallel plate capacitor, whose capacitance increases with decreasing distance between the plates. But this effect does not account for all the change in capacitance observed. In fact, most of it is due to the gradual softening of the coupled electromechanical system. This effect leads to a larger structural response for a given voltage increment at higher bias, which in turn means that more charge must be added to retain the voltage difference between the electrodes.

Reference

1. R.K. Gupta, *Electrostatic Pull-In Structure Design for In-Situ Mechanical Property Measurements of Microelectromechanical Systems (MEMS)*, Ph.D. thesis, MIT, 1997.

Application Library path: `MEMS_Module/Actuators/electrostatically_actuated_cantilever`

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

Use microns to define the geometry units.

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**.

   Create the geometry so that a swept mesh can be used subsequently. Three blocks are required as to sweep the mesh no change in the *Y*-*Z* cross section is allowed.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
3 In the **Width** text field, type 320.
4 In the **Depth** text field, type 10.
5 In the **Height** text field, type 2.
6 Locate the **Position** section. In the **z** text field, type 2.
7 Right-click **Block 1 (blk1)** and choose **Build Selected**.

Block 2 (blk2)

1 On the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
3 In the **Width** text field, type 320.

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

5 In the **Height** text field, type 24.

Block 3 (blk3)

1 On the **Geometry** toolbar, click **Block**.

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

3 In the **Width** text field, type 300.

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

5 In the **Height** text field, type 24.

6 Click **Build All Objects**.

Add a parameter for the DC voltage applied to the cantilever.

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:

Name	Expression	Value	Description
V0	5[V]	5 V	Bias on cantilever

The cantilever is assumed to be heavily doped so that it acts as a conductor, held at constant potential. The **Linear Elastic Material** feature is therefore used.

ELECTROMECHANICS (EMI)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromechanics (emi)** and choose **Linear Elastic Material**.

2 Select Domain 2 only.

Fix one end of the cantilever.

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Electromechanics (emi)** and choose the boundary condition **Structural>Fixed Constraint**.

2 Select Boundary 4 only.

Since only half of the cantilever is included in the model, the symmetry condition should be applied on the mid-plane of the solid. The electric field default condition (**Zero Charge**) is equivalent to a symmetry condition, so only the structural symmetry boundary condition needs to be applied.

Symmetry 1

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Structural> Symmetry**.

2 Select Boundary 5 only.

Use the **Terminal** feature to set the voltage on the exterior of the cantilever.

Terminal 1

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Electrical> Terminal**.

2 Select Boundaries 6, 8, 10, and 15 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 `V0`.

Set up the ground plane underneath the cantilever.

Ground 1

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Electrical> Ground**.

2 Select Boundaries 3 and 14 only.

Apply boundary conditions to constrain the mesh deformation.

Prescribed Mesh Displacement 2

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Deformed Mesh> Prescribed Mesh Displacement**.

2 Select Boundaries 2, 7, 13, 16, 18, 23, and 24 only.

3 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Prescribed Mesh Displacement** section.

4 Clear the **Prescribed z displacement** check box.

This way, you allow the mesh nodes to move in the z direction while they are fixed in the x and y directions.

Confirm that the default features have acquired the correct selections.

5 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

Prescribed Mesh Displacement 1

The default mesh displacement feature constrains all the remaining boundaries to have zero displacement.

Electromechanical Interface 1

The **Electromechanical Interface** feature applies forces to the exterior boundaries of the cantilever.

Zero Charge 1

The **Zero Charge** feature applies symmetry conditions to the remaining boundaries where the electric potential is solved for.

Add Materials to the model.

MATERIALS

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	4.5	1	Basic
Young's modulus	E	153[GPa]	Pa	Basic
Poisson's ratio	nu	0.23	1	Basic
Density	rho	2330	kg/m³	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 Select Domains 1, 3, and 4 only.

Set the material to be non-solid, to ensure the interface solves the electrostatics equations in the spatial frame.

3 In the **Settings** window for **Material**, click to expand the **Material properties** section.

4 Locate the **Material Properties** section. From the **Material type** list, choose **Nonsolid**.

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

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	1	1	Basic

Create a swept mesh.

MESH 1

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Mapped**.

2 Select Boundary 4 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Edge 5 only.

Size

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

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

3 From the **Predefined** list, choose **Extremely fine**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 4.

Free Quad 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Free Quad**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Free Quad**, click **Build All**.

Swept 1

1 Right-click **Mesh 1** and choose **Swept**.

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

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

4 Select Domains 1 and 2 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Swept 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `20`.

4 Click **Build All**.

Swept 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

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

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

4 Select Domains 3 and 4 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Swept 2** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, click **Build All**.

Set up a **Parametric Sweep** over the applied voltage.

STUDY 1

Step 1: Stationary

1 In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.

2 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

3 Click **Add**.

4 Click **Range**.

5 In the **Range** dialog box, type `1` in the **Start** text field.

6 In the **Step** text field, type `1`.

7 In the **Stop** text field, type `6`.

8 Click **Add**.

Add points at 6.05 and 6.1 V to the sweep by adding these points after the range statement. The table field should now contain: `range(1,1,6) 6.05 6.1`.

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

RESULTS

Displacement (emi)

Create additional data sets for post processing. First create a mirrored data set.

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 **Plane Data** section.

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

Then create a data set with some surface selections.

Selection

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

2 On the **Results** toolbar, click **Selection**.

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

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

5 Select Boundaries 3–6, 8, 10, 14, and 15 only.

Finally, mirror this data set.

Mirror 3D 2

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 1/Solution 1 (2) (sol1)**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

Edit the default displacement plot to show the z-displacement and the corresponding mesh deformation.

Displacement (emi)

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

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

3 From the **Data set** list, choose **Mirror 3D 1**.

Surface 1

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

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> Electromechanics (Solid Mechanics)>Displacement> Displacement field (material and geometry frames)>w - Displacement field, Z component**.

3 Locate the **Coloring and Style** section. Select the **Reverse color table** check box.

Volume 1

1 In the **Model Builder** window, under **Results** right-click **Displacement (emi)** and choose **Volume**.
2 In the **Settings** window for **Volume**, locate the **Data** section.
3 From the **Data set** list, choose **Study 1/Solution 1 (1) (sol1)**.
4 Locate the **Expression** section. In the **Expression** text field, type `z-Z`.
5 Click to expand the **Shrink elements** section. Locate the **Shrink Elements** section. In the **Element scale factor** text field, type `0.8`.
6 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Slice 1

1 Right-click **Displacement (emi)** and choose **Slice**.
2 In the **Settings** window for **Slice**, locate the **Expression** section.
3 In the **Expression** text field, type `z-Z`.
4 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.
5 On the **Displacement (emi)** toolbar, click **Plot**.

Edit the default potential plot.

Potential (emi)

1 In the **Model Builder** window, under **Results** click **Potential (emi)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Data set** list, choose **Mirror 3D 1**.

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 In the **Planes** text field, type `7`.

Surface 1

1 In the **Model Builder** window, under **Results** right-click **Potential (emi)** 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>Electromechanics (Electrical Quasistatics)>Electric>V - Electric potential**.

3 Locate the **Data** section. From the **Data set** list, choose **Mirror 3D 2**.

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

Arrow Volume 1

1 Right-click **Potential (emi)** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Electromechanics (Electrical Quasistatics)>Electric>emi.Ex,...,emi.Ez - Electric field (spatial frame)**.

3 Locate the **Arrow Positioning** section. Find the **y grid points** subsection. In the **Points** text field, type `15`.

4 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Normalized**.

5 On the **Potential (emi)** toolbar, click **Plot**.

Add a plot to show the deformed shape of the underside of the cantilever.

Line Graph 1

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

2 In the **Model Builder** window, right-click **1D Plot Group 3** and choose **Line Graph**.

3 Select Edge 6 only.

4 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> Electromechanics (Solid Mechanics)>Displacement> Displacement field (material and geometry frames)>w - Displacement field, Z component**.

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

1D Plot Group 3

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Legend** section.

3 From the **Position** list, choose **Lower left**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Shape of cantilever displacement for different applied voltages`.

6 Right-click **Results>1D Plot Group 3** and choose **Rename**.

7 In the **Rename 1D Plot Group** dialog box, type `Displacement vs Applied Voltage` in the **New label** text field.

8 Click **OK**.

9 On the **Displacement vs Applied Voltage** toolbar, click **Plot**.

Add a plot of tip displacement vs applied DC voltage.

Point Graph 1

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

2 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Point Graph**.

3 Select Point 10 only.

4 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1> Electromechanics (Solid Mechanics)>Displacement> Displacement field (material and geometry frames)>w - Displacement field, Z component**.

1D Plot Group 4

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 4** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Tip Displacement vs Applied Voltage` in the **New label** text field.

3 Click **OK**.

4 On the **Tip Displacement vs Applied Voltage** toolbar, click **Plot**.

Finally, plot the DC capacitance of the device vs voltage.

Global 1

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

2 In the **Model Builder** window, right-click **1D Plot Group 5** and choose **Global**.

3 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 - Maxwell capacitance**.

Modify the automatically generated expression to account for the symmetry boundary condition.

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

Expression	Unit	Description
2*emi.C11	fF	Capacitance

1D Plot Group 5

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 5** and choose **Rename**.
2 In the **Rename 1D Plot Group** dialog box, type `DC C-V Curve` in the **New label** text field.
3 Click **OK**.
4 On the **DC C-V Curve** toolbar, click **Plot**.

Gecko Foot

Introduction

In nature, various species apply advanced techniques for specialized tasks. For instance, gecko lizards use dry adhesion forces such as van der Waals forces to climb walls. Dry adhesion is a phenomenon of interest for sticking because it requires no energy to hold on, and no residue is left on the surface. Gecko lizards have inspired researchers to develop synthetic gecko foot hairs for use in, for example, robots.

A strand of hair on a gecko foot is a very complex biological structure with hierarchical nanosections and microsections. On its feet, a gecko has billions of nanoscale hairs that are in contact with surfaces while it climbs. These nanohairs are attached to microscale hairs, which are on the tip of the gecko's toes.

Critical design parameters for nanohairs to achieve optimal sticking are hair length, detach angle, distance between nanohairs, and the cross-sectional area of a single strand of hair. By varying these parameters, it is possible to design hairs that can stick to very rough surfaces. At the same time, they must be stiff enough to avoid sticking to each other. Proper material choices help achieving the design goals while providing the required adhesion force. Typically the Young's modulus for materials used in synthetic nanohair vary between 1 GPa and 15 GPa.

Model Definition

This model contains the hierarchy of synthetic gecko foot hair where nanoscale and microscale cantilever beams describe the seta and spatula parts of a spatular stalk attached to a gecko foot. The basis of the analyzed structure is a microscale stalk with the following dimensions: width, 4.53 μm; height, 4.33 μm; and length, 75 μm. At the end of the microhair, 169 nanohairs are attached and they have dimensions of 0.18 μm, 0.17 μm, and 3 μm, respectively. The microhair is fixed at the far end, while the contact and friction forces appear as surface loads at the end of each nanohair. The free-body diagram of a micro/nanohair in Figure 1 illustrates the applied forces, which are set to 0.4 μN for the contact force and 0.2 μN for the friction force with 60° contact angle to target surface. The structure is made of β-keratin with a Young's modulus of 2 GPa and a Poisson's ratio of 0.4. The model was inspired by Ref. 1.

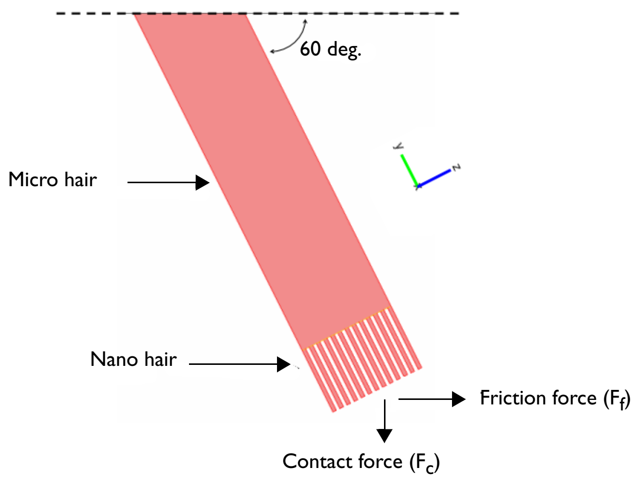


Figure 1: Model of the tip of a gecko foot hair.

Results and Discussion

The plot in Figure 2 shows the von Mises effective stress in the model. Table 1 lists the maximum values of the von Mises stress, total displacement, and principal strain:

TABLE 1: STRESS, DISPLACEMENT, AND PRINCIPAL STRAIN RESULTS IGNORING GEOMETRIC NONLINEARITY.

MAXIMUM VON MISES STRESS (N/m^2)	MAXIMUM TOTAL DISPLACEMENT (µm)	MAXIMUM FIRST PRINCIPAL STRAIN
$9.52 \cdot 10^7$	12.6	0.0505

The maximum von Mises stress in the analyzed model is almost twice the value of the material's yield stress, which clearly indicates that further investigation is required.

The deformed plot in Figure 2 shows that displacement is large, while the results in the table above indicate that the maximum strain is moderate. Hence, the next step is to enable the geometric nonlinearity within the linear elastic material model. (An alternative would be to search for a suitable hyperelastic material model, which would require the corresponding material data.)

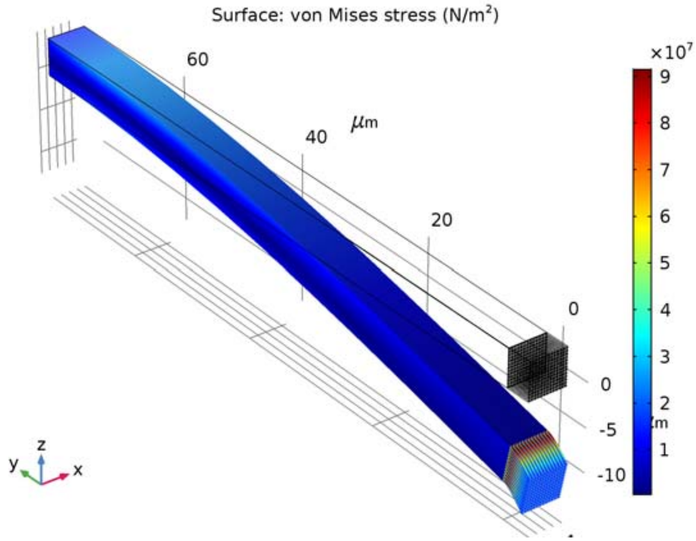


Figure 2: Deformed shape plot of the von Mises stress in a synthetic gecko foot ignoring geometric nonlinearity. The plot shows the displacements without any scaling.

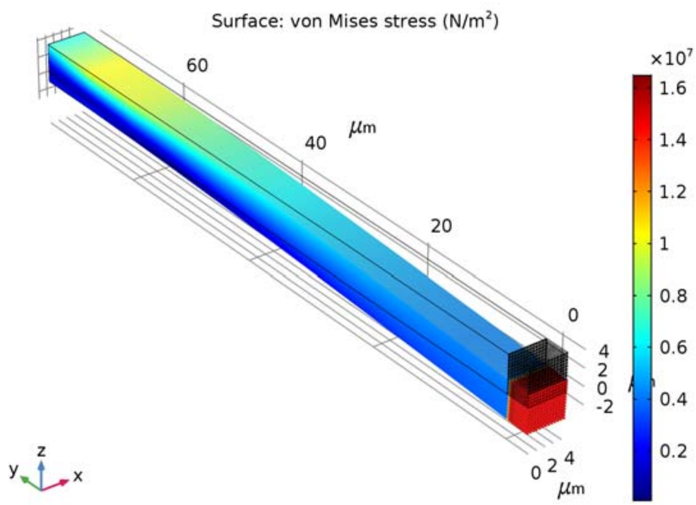


Figure 3: The result recomputed with the geometric nonlinearity taken into account.

You include the geometric nonlinearity by selecting the corresponding check box in the Linear Elastic Material Model feature node. The resulting changes in the model equations are shown under the Equation section therein.

The plot in Figure 3 shows the resulting von Mises stress distribution and the deformation, and Table 2 below gives the recomputed maximum values.

TABLE 2: STRESS, DISPLACEMENT, AND PRINCIPAL STRAIN RESULTS INCLUDING GEOMETRIC NONLINEARITY EFFECTS.

MAXIMUM VON MISES STRESS (N/m^2)	MAXIMUM TOTAL DISPLACEMENT (μm)	MAXIMUM FIRST PRINCIPAL STRAIN
$1.73 \cdot 10^7$	2.93	$8.85 \cdot 10^{-3}$

The results show that without the geometric nonlinearity taken into account, the model overpredicts the maximum von Mises stress by more than a factor of five, and the maximum displacement by more than a factor of four. Furthermore, the maximum strain computed with the geometric nonlinearity included becomes less than 1%, which eliminates the need of further analysis involving more complicated hyperelastic material models.

References

1. G. Shah and I. Lee, *Finite Element Analysis of Gecko Foot Hairs for Dry Adhesive Design and Fabrication*, Dept. of Mechanical Engineering—NanoRobotics Lab, Carnegie Mellon Univ., Pittsburgh.

2. M. Sitti and R.S. Fearing, "Synthetic Gecko Foot-Hair Micro/Nano-Structures for Future Wall-Climbing Robots," *Proc. IEEE Robotics and Automation Conf.*, Sept. 2003.

3. J. Vincent, *Structural Biomaterials*, rev. ed., Princeton University Press, 1990.

Application Library path: `MEMS_Module/Actuators/gecko_foot`

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>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

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:

Name	Expression	Value	Description
Fc	0.4[uN]	4E-7 N	Contact force
Ff	0.2[uN]	2E-7 N	Friction force
theta	pi/3	1.047	Contact angle
Dm	75[um]	7.5E-5 m	Microhair length
Hm	4.33[um]	4.33E-6 m	Microhair width
Wm	4.53[um]	4.53E-6 m	Microhair height
Dn	3[um]	3E-6 m	Nanohair length
Hn	0.17[um]	1.7E-7 m	Nanohair width
Wn	0.18[um]	1.8E-7 m	Nanohair height
Area	Wn*Hn	3.06E-14 m^2	Cross-sectional area of the spatulae

GEOMETRY 1

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**.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

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

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

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

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

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

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

8 In the **Rename Block** dialog box, type `Nanohair` in the **New label** text field.

9 Click **OK**.

Next, add object selections for the nanohair's root and end. An object selection is defined by a set of geometric entities at a specified level selected from the geometric objects that precede the object selection feature in the geometry sequence. Their main advantage compared to regular selection features is that object selections propagate through the geometry sequence, which make them robust and convenient to apply in cases like the present one.

Explicit Selection 1 (sel1)

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

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

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

4 On the object **blk1**, select Boundary 3 only.

5 Right-click **Explicit Selection 1 (sel1)** and choose **Rename**.

6 In the **Rename Explicit Selection** dialog box, type `Nanohair ends` in the **New label** text field.

The use of plural anticipates the fact that you will create an array of nanohairs; this object selection will automatically propagate to all of these.

7 Click **OK**.

Explicit Selection 2 (sel2)

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

2 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

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

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

5 On the object **blk1**, select Boundary 6 only.

6 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

7 Right-click **Explicit Selection 2 (sel2)** and choose **Build Selected**.

8 Right-click **Explicit Selection 2 (sel2)** and choose **Rename**.

9 In the **Rename Explicit Selection** dialog box, type `Nanohair roots` in the **New label** text field.

10 Click **OK**.

Array 1 (arr1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **blk1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `13`.

5 In the **z size** text field, type `13`.

6 Locate the **Displacement** section. In the **x** text field, type `(Wm-Wn)/12`.

7 In the **z** text field, type `(Hm-Hn)/12`.

8 Right-click **Array 1 (arr1)** and choose **Build Selected**.

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

Block 2 (blk2)

1 On the **Geometry** toolbar, click **Block**.

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

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

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

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

6 Right-click **Block 2 (blk2)** and choose **Build Selected**.

7 Right-click **Block 2 (blk2)** and choose **Rename**.

8 In the **Rename Block** dialog box, type `Microhair` in the **New label** text field.

9 Click **OK**.

Explicit Selection 3 (sel3)

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

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

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

4 On the object **blk2**, select Boundary 3 only.

5 Right-click **Explicit Selection 3 (sel3)** and choose **Build Selected**.

6 Right-click **Explicit Selection 3 (sel3)** and choose **Rename**.

7 In the **Rename Explicit Selection** dialog box, type `Microhair end` in the **New label** text field.

8 Click **OK**.

9 In the **Settings** window for **Explicit Selection**, click **Go to Default View**.

Form Union (fin)

Use an assembly to connect parts of the geometry of significantly different dimensions.

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

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Clear the **Create pairs** check box.

5 Right-click **Component 1 (comp1)>Geometry 1>Form Union (fin)** and choose **Build Selected**.

The model geometry is now complete.

DEFINITIONS

Because of the use of assembly, you need to add an identity pair to set up the displacement field continuity at the interfaces, where the nanohairs are attached to the microhair. Configure the identity to operate on the material frame, because this is the frame used within the **Solid Mechanics** interface.

Identity Boundary Pair 1 (p1)

1 On the **Definitions** toolbar, click **Pairs** and choose **Identity Boundary Pair**.

2 In the **Settings** window for **Pair**, locate the **Source Boundaries** section.

3 From the **Selection** list, choose **Microhair end**.

4 Locate the **Destination Boundaries** section. Select the **Active** toggle button.

5 From the **Selection** list, choose **Nanohair roots**.

Define a rotated coordinate system to account for the contact angle.

Rotated System 2 (sys2)

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

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

3 Find the **Euler angles (Z-X-Z)** subsection. In the β text field, type `pi/2-theta`.

MATERIALS

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	2e9	Pa	Basic
Poisson's ratio	nu	0.4	1	Basic
Density	rho	1200	kg/m³	Basic

DEFINITIONS

Set up a maximum operator to compute maximum values of the von Mises stress, total displacement, and principal strain over the entire geometry.

Maximum 1 (maxop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Maximum**.

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

3 From the **Selection** list, choose **All domains**.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
max_v_Mises	maxop1(solid.mises)	N/m²	Maximum von Mises stress
max_disp	maxop1(solid.disp)	m	Maximum total displacement
max_ep1	maxop1(solid.ep1)		Maximum first principal strain

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

2 Select Boundary 83 only.

Boundary Load 1

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Boundary Load**.

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

3 From the **Selection** list, choose **Nanohair ends**.

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	x1
-Fc/Area	x2
Ff/Area	x3

Continuity 1

1 Right-click **Solid Mechanics (solid)** and choose **Pairs>Continuity**.

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

3 In the **Pairs** list, select **Identity Boundary Pair 1 (p1)**.

MESH 1

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Mapped**.

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

3 From the **Selection** list, choose **Nanohair ends**.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 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 `0.1`.

Mapped 1

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

2 In the **Settings** window for **Mapped**, click **Build Selected**.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

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

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

4 From the **Selection** list, choose **All domains**.

5 Remove Domain 14 from the selection.

6 Right-click **Component 1 (comp1)>Mesh 1>Swept 1** and choose **Distribution**.

7 In the **Settings** window for **Swept**, click **Build Selected**.

Mapped 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Mapped**.

2 Select Boundary 83 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 2** and choose **Distribution**.

2 Select Edges 162–164 and 168 only.

Mapped 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Mapped 2**.

2 In the **Settings** window for **Mapped**, click **Build Selected**.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 Right-click **Swept 2** and choose **Distribution**.

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

The mesh is now complete.

STUDY 1

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

RESULTS

Stress (solid)

Reproduce the plot in by following these steps:

1 In the **Model Builder** window, expand the **Stress (solid)** node.

Deformation

1 In the **Model Builder** window, expand the **Results>Stress (solid)>Surface 1** node, then click **Deformation**.
2 In the **Settings** window for **Deformation**, locate the **Scale** section.
3 Select the **Scale factor** check box.
4 In the associated text field, type 1.
5 On the **Stress (solid)** toolbar, click **Plot**.
6 Click the **Zoom Extents** button on the **Graphics** toolbar.

   Compare the resulting plot to that in .

Derived Values

Next, calculate the maximum values for von Mises stress, total displacement, and first principal strain; compare the results to those shown in .

Global Evaluation 1

1 On 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>Definitions>Variables>max_v_Mises - Maximum von Mises stress**.
3 Click **Evaluate**.

Global Evaluation 2

1 On 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>Definitions>Variables>max_disp - Maximum total displacement**.
3 Click **Table 1 - Global Evaluation 1 (max_v_Mises)**.

Global Evaluation 3

1 On 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>Definitions> Variables>max_ep1 - Maximum first principal strain**.

3 Click **Table 1 - Global Evaluation 1 (max_v_Mises)**.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

Now, switch on geometric nonlinearity.

STUDY 1

Step 1: Stationary

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

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

3 Select the **Include geometric nonlinearity** check box.

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

RESULTS

Stress (solid)

Compare the plot in the Graphics window with that in .

Global Evaluation 1

Finally, compute the new maximum values and display them in a new table. Compare the results to those in .

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 1**.

2 In the **Settings** window for **Global Evaluation**, click **New Table**.

Global Evaluation 2

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 2**.

2 In the **Settings** window for **Global Evaluation**, click **Table 2 - Global Evaluation 1 (max_v_Mises)**.

Global Evaluation 3

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 3**.

2 In the **Settings** window for **Global Evaluation**, click **Table 2 - Global Evaluation 1 (max_v_Mises)**.

Created in COMSOL Multiphysics 5.3

Thermal Stresses in a Layered Plate

Introduction

This example contains an analysis of the thermal stress in a layered plate. The plate consists of three layers: the coating, the substrate, and the carrier. The coating is deposited on the substrate at a temperature of 800 °C. At this temperature both the coating and the substrate are stress-free. The temperature of the plate is then lowered to 150 °C, which induces thermal stresses in the coating/substrate assembly. At this temperature the coating/substrate assembly is epoxied to a carrier plate so that the coating/substrate has initial stresses when it is bonded to the carrier. Finally, the temperature is lowered to 20 °C.

Model Definition

The plate is restrained from moving in the z direction. This makes it possible to use the plane strain approximation in the 2D Solid Mechanics interface with. The assumption is then that the z-component of the strain is zero.

This model contains only thermal loads, which are introduced into the constitutive equations according to the following equations:

$$\sigma = D\varepsilon_{\mathrm{el}} + \sigma_0 = D(\varepsilon - \varepsilon_{\mathrm{th}} - \varepsilon_0) + \sigma_0$$

and

$$\varepsilon_{\mathrm{th}} = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix}_{\mathrm{th}} = \alpha_{\mathrm{vec}}(T - T_{\mathrm{ref}})$$

where σ is the stress vector, D is the elasticity matrix, ε_x, ε_y, ε_z, γ_{xy}, γ_{yz}, γ_{xz} are the strain components, α_{vec} is the coefficient of thermal expansion, T is the actual temperature, and T_{ref} is the reference temperature.

The geometry of the plate is shown in Figure ???. The top layer in the geometry is the coating, the middle layer is the substrate, and the bottom layer is the carrier.

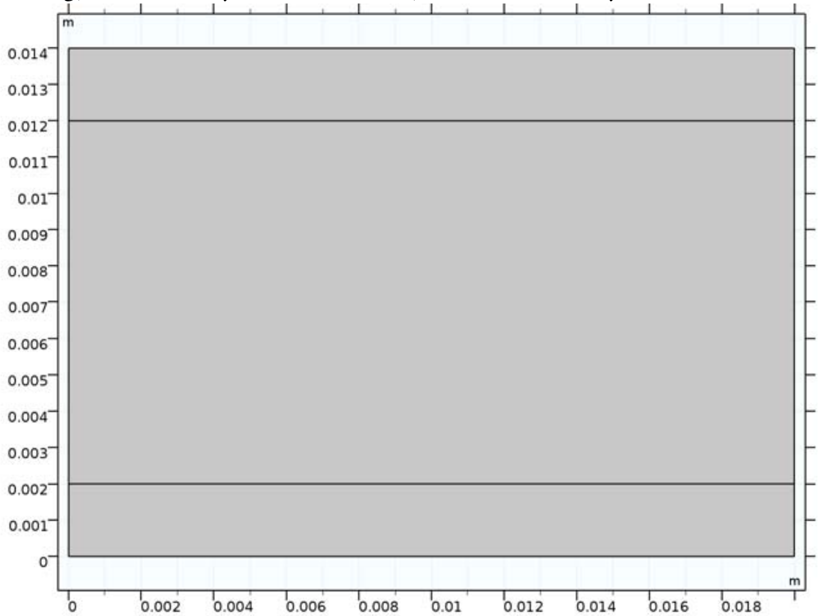


Figure 1: The plate geometry.

The analysis uses two steps:

STEP 1

In the first step you lower the temperature from 800 °C to 150 °C, which affects the coating layer and the substrate layer. The carrier layer is not active in this step.

In both steps the upper-left corner of the coating is fixed, and the upper-right corner of the coating is constrained in the *y* direction. This prevents rigid-body movements but does not affect the stress distribution.

STEP 2

In this step all three layers are active and you drop the temperature from 150 °C to room temperature, 20 °C. This step includes the initial stresses from Step 1.

Results and Discussion

Figure ??? depicts the normal stress in the x direction from the first analysis step. The substrate material has a higher coefficient of thermal expansion than the coating material. This means that the substrate shrinks more than the coating, causing tensile stresses in the substrate area next to the coating and compressive stresses in the coating.

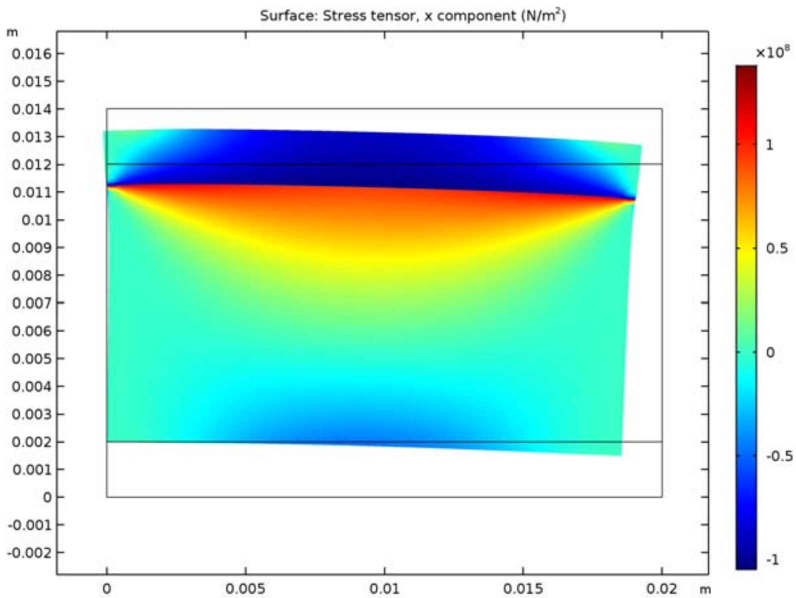


Figure 2: Normal stress in the x direction for the first analysis step.

Figure ??? shows the residual thermal x-stress in the final step where the temperature is lowered to 20 °C. The tensile stress levels have increased somewhat in the substrate area next to the coating, as have the compressive stress in the coating compared to the first process step. The main stress contribution is clearly the added initial stress from the first process step.

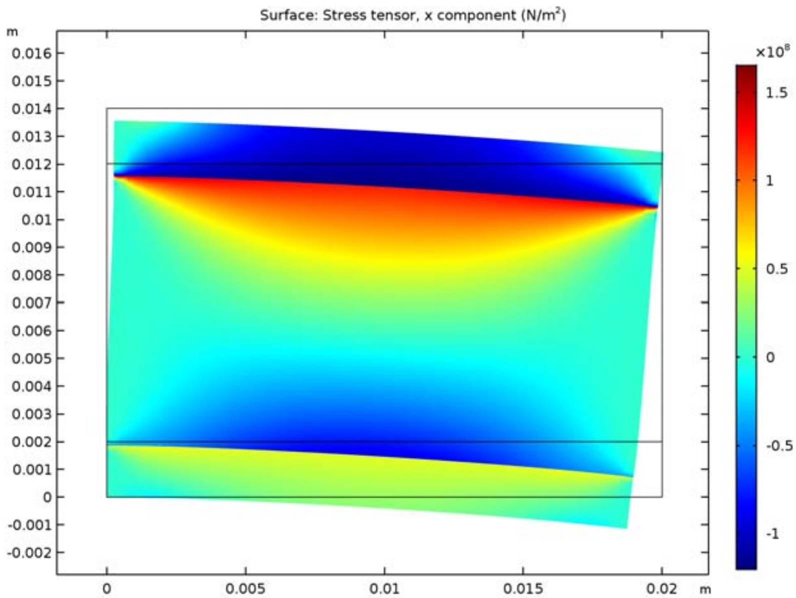


Figure 3: Residual thermal stress at room temperature.

Application Library path: `MEMS_Module/Actuators/layered_plate`

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 **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

Add two different solid interfaces, one for the structure before adding the carrier, and one for the complete structure.

3 Click **Add**.

4 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

8 Click **Done**.

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:

Name	Expression	Value	Description
`Ttop`	`800[degC]`	1073 K	`Coating deposition temperature`
`Tbot`	`150[degC]`	423.2 K	`Temperature when the coating/ substrate is epoxied to the carrier`
`Troom`	`20[degC]`	293.2 K	`Room temperature`

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Height** text field, type `0.002`.

4 In the **Width** text field, type `0.02`.

5 Click **Build All Objects**.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.02`.

4 In the **Height** text field, type `0.01`.

5 Locate the **Position** section. In the **y** text field, type `0.002`.

6 Click **Build All Objects**.

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.02`.

4 In the **Height** text field, type `0.002`.

5 Locate the **Position** section. In the **y** text field, type `0.012`.

6 Click **Build All Objects**.

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

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, type `Two Layers` in the **Label** text field.

3 Select Domains 2 and 3 only.

TWO LAYERS (SOLID)

On the **Physics** toolbar, click **Solid Mechanics (solid)** and choose **Two Layers (solid)**.

Linear Elastic Material 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Two Layers (solid)** node.

Thermal Expansion 1

1 Right-click **Linear Elastic Material 1** and choose **Thermal Expansion**.

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

3 In the T_{ref} text field, type `Ttop`.

4 Locate the **Model Input** section. In the T text field, type `Tbot`.

Rigid Motion Suppression 1

1 In the **Model Builder** window, right-click **Two Layers (solid)** and choose **Domain Constraints>Rigid Motion Suppression**.

2 In the **Settings** window for **Rigid Motion Suppression**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

SOLID MECHANICS 2 (SOLID2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics 2 (solid2)**.

2 In the **Settings** window for **Solid Mechanics**, type `Three Layers` in the **Label** text field.

THREE LAYERS (SOLID2)

On the **Physics** toolbar, click **Solid Mechanics 2 (solid2)** and choose **Three Layers (solid2)**.

Thermal Expansion 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Three Layers (solid2)** right-click **Linear Elastic Material 1** and choose **Thermal Expansion**.
2. In the **Settings** window for **Thermal Expansion**, locate the **Thermal Expansion Properties** section.
3. In the T_{ref} text field, type `Tbot`.
4. Locate the **Model Input** section. In the T text field, type `Troom`.

Rigid Motion Suppression 1

1. In the **Model Builder** window, right-click **Three Layers (solid2)** and choose **Domain Constraints>Rigid Motion Suppression**.
2. In the **Settings** window for **Rigid Motion Suppression**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **All domains**.

Linear Elastic Material 1

Use the stresses from the two layer model as initial stresses for the three layer model.

Initial Stress and Strain 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Three Layers (solid2)** right-click **Linear Elastic Material 1** and choose **Initial Stress and Strain**.
2. Select Domains 2 and 3 only.
3. In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.
4. In the S_0 table, enter the following settings:

`solid.sx`	`solid.sxy`	0
solid.sxy	`solid.sy`	0
0	0	`solid.sz`

MATERIALS

Material 1 (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, type `Carrier` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Manual**.

4 Click **Clear Selection**.

5 Select Domain 1 only.

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

Property	Name	Value	Unit	Property group
Young's modulus	E	`2.15e11`	Pa	Basic
Poisson's ratio	nu	`0.3`	1	Basic
Density	rho	`1000`	kg/m³	Basic
Coefficient of thermal expansion	alpha	`6e-6`	1/K	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

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

3 Select Domain 2 only.

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

Property	Name	Value	Unit	Property group
Young's modulus	E	`1.3e11`	Pa	Basic
Poisson's ratio	nu	`0.28`	1	Basic
Density	rho	`1000`	kg/m³	Basic
Coefficient of thermal expansion	alpha	`3e-6`	1/K	Basic

Material 3 (mat3)

1 Right-click **Materials** and choose **Blank Material**.

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

3 Select Domain 3 only.

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

Property	Name	Value	Unit	Property group
Young's modulus	E	`7e10`	Pa	Basic
Poisson's ratio	nu	`0.17`	1	Basic

Property	Name	Value	Unit	Property group
Density	rho	1000	kg/m³	Basic
Coefficient of thermal expansion	alpha	5e-7	1/K	Basic

MESH 1

Size

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Extra fine**.
4. Click **Build All**.

STUDY 1

Add a static solution for the case with three layers.

Step 2: Stationary 2

1. On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

   Use only one Solid Mechanics interface per solution by deactivating the other one.

Step 1: Stationary

1. In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2. In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3. In the table, clear the **Solve for** check box for **Three Layers**.

Step 2: Stationary 2

1. In the **Model Builder** window, under **Study 1** click **Step 2: Stationary 2**.
2. In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3. In the table, clear the **Solve for** check box for **Two Layers**.
4. On the **Study** toolbar, click **Compute**.

RESULTS

Surface 1

1. In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.
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>Two Layers>Stress> Stress tensor (spatial frame)>solid.sx - Stress tensor, x component**.

3 On the **Stress (solid)** toolbar, click **Plot**.

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

Surface 1

1 In the **Model Builder** window, expand the **Results>Stress (solid2)** node, then click **Surface 1**.

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>Three Layers>Stress> Stress tensor (spatial frame)>solid2.sx - Stress tensor, x component**.

3 On the **Stress (solid2)** toolbar, click **Plot**.

Micropump Mechanism[1]

1. This model is courtesy of Matthew J. Hancock and Stuart Brown of Veryst Engineering, LLC.

Micropumps are key components of microfluidic systems with applications ranging from biological fluid handling to microelectronic cooling. This model simulates the mechanism of a valveless micropump, that is designed to be effective at low Reynolds numbers, overcoming hydrodynamic reversibility. Valveless pumps are often preferred in microfludic systems because they minimize the risk of clogging and are gentle on biological material. The Fluid-Structure Interaction interface is used to solve for the flow of the fluid and the associated deformation of the structure. In addition the Global ODEs and DAEs interface is used to demonstrate how to perform a time resolved integration of the total flow throughout the pumping cycle.

Introduction

Many valveless pump designs are ineffective when the system has a low Reynolds number, and consequently are unsuitable for viscous fluids and applications with small length scales or low flow rates. This is largely because without valves it is difficult to achieve sustained flow in a given direction.

The mechanism simulated in this model overcomes this limitation by converting oscillatory fluid motion, induced by a simple reciprocating pumping mechanism, into a net flow in one direction. It is relatively easy to create an oscillatory pumping mechanism in a microfluidic system, e.g. a membrane can be vibrated by a piezo oscillator to periodically vary the volume of a microchamber. In this model, an oscillatory flow is fed into a channel containing bendable microflaps. The deformation of the microflaps in response to the motion of the fluid alters the flow and results in a net flow rate in a consistent direction. The passive nature of the flow regulator allows for directional control of the flow without the use of the complicated synchronized actuation mechanisms that would be required in a valve based system.

In this model the Fluid-Stricture Interaction interface is used to specify the input oscillatory flow, along with the mechanical properties of the flaps. In addition, some best practice guidelines for ensuring good mesh deformation in moving-mesh problems are introduced. The deformation of the flaps, and the flow of the fluid, is calculated as a function of time for two full cycles of the pumping mechanism. This allows the physical mechanism responsible for generating the unidirectional flow to be visualized clearly using an animation. As well as visualizing flow rate and direction as a function of time throughout the pumping cycle, integration coupling components are used in conjunction with the Global ODEs and DAEs interface to calculate the net volumed pumped from left-to-right as a function of time. This is an example of how the functionality of one COMSOL interface can be enhanced by using a custom equation specified in a mathematics interface, and demonstrates the ease with which user defined equations can be incorporated into COMSOL models.

Model Definition

The model geometry is shown in Figure 1. It consists of a horizontal channel that is 600 μm in length and 100 μm high. A vertical chamber connects to the channel at the midpoint along its length. Two tilted flaps are attached to the bottom of the channel such that they partially obstruct flow along the channel length. The are spaced to be centered on the midpoint of the channel length, and they are both angled at 45 degrees to the to the horizontal channel edge. Note that this 2D model represents a cross-section through the midpoint of the channel in the out-of-plane direction. An out-of-plane thickness of 10 μm has been used for the purpose of calculating the volume of fluid pumped as a function of time. However, as no edge effects due to walls that are out-of-plane are included, this is equivalent to modeling a 10 μm deep section of a much thicker channel.

Note the vertical lines that connect the apex of the tilted flaps to the top of the channel. These are included as guidelines to aid the meshing algorithm as the structure deforms. Point Integration coupling components are applied to the apex of each bendable flap. These are then used to compute the horizontal displacement of the flap apex by calculating the position of the apex relative to the undeformed geometry. This is achieved by evaluating the expression `(x-X)` using the point Integration coupling components, where `x` and `X` are the x-coordinate of the apex point in the undeformed and deformed geometry, respectively. Thus, `(x-X)` gives the horizontal shift experienced by the vertical guidelines. Prescribed Mesh Displacement features are applied to the guidelines to force the x-displacement of the mesh along the guidelines to be equal to that of the flap apexes. This helps the moving mesh, which is automatically recalculated as the structure deforms, to maintain a reasonable structure around the bending flaps. In general, it is good practice to take measures like this to ensure that the mesh deforms in an appropriate way.

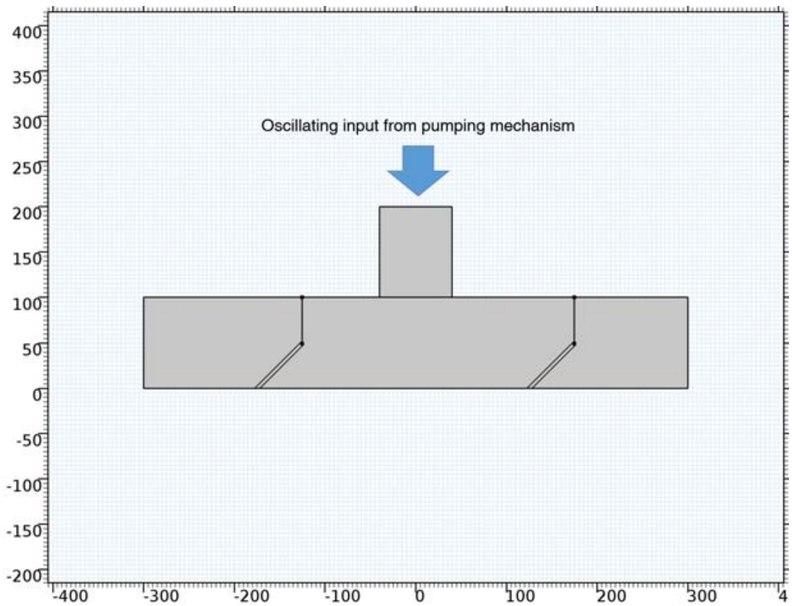


Figure 1: The model geometry consists of a horizontal channel and a vertical chamber. Tilted flaps are positioned within the channel, the response of these flaps to the oscillatory fluid motion induced via the labeled boundary results in a net flow from left-to-right.

The physics required for the model is configured within the Fluid-Structure Interaction interface. The Linear Elastic Material feature is applied to the two tilted flaps, and then the Fluid-Solid Interface Boundary feature is automatically assigned to the boundaries between the flaps and the fluid in the channel. An Inlet boundary feature is applied to the top of the vertical chamber. This specifies the inflow velocity, via a user input expression, to vary sinusoidally in time with a period of 1 s. An Outlet boundary feature is applied to the left and right boundaries of the channel. Two boundary integration coupling components, named `intopL()` and `intopR()` for the left and right outlets respectively, are also applied to these outlet boundaries. These are used to compute the flow rate out of each outlet. This is achieved using some user defined variables in the Definitions node within Component 1. The flow rate from each outlet is calculated by integrating the depended variable `u_fluid`, which is the horizontal component of the fluid velocity, and multiplying by the out-of-plane length scale of 10 µm. The net flow rate out of the channel, `UoutNet`, is then calculated from the difference between the flow from the left and right outlets, such that positive values correspond to a net flow in the left-to-right direction.

A Global ODEs and DAEs interface is added to compute the integrated net flow as a function of time. This is achieved using a Global Equation which integrates `UoutNet` with respect to time to obtain `Vpump`. This step is necessary as `UoutNet` gives the instantaneous net flow rate as a function in time, however a more useful metric for evaluating a pump is the total volume pumped throughout an entire pumping cycle. Note that the `timeint()` operator can also be used to visualize the time integration of a variable. However, use of `timeint()` is not as accurate as directly solving for an integrated quantitiy, as the `timeint()` operator only uses the timesteps which are saved in the solution but solving directly uses every timestep taken by the solver.

The mesh is configured to be tightest around the tilted flaps, in order to resolve the stress within the bending flaps. The mesh is shown in Figure 2.

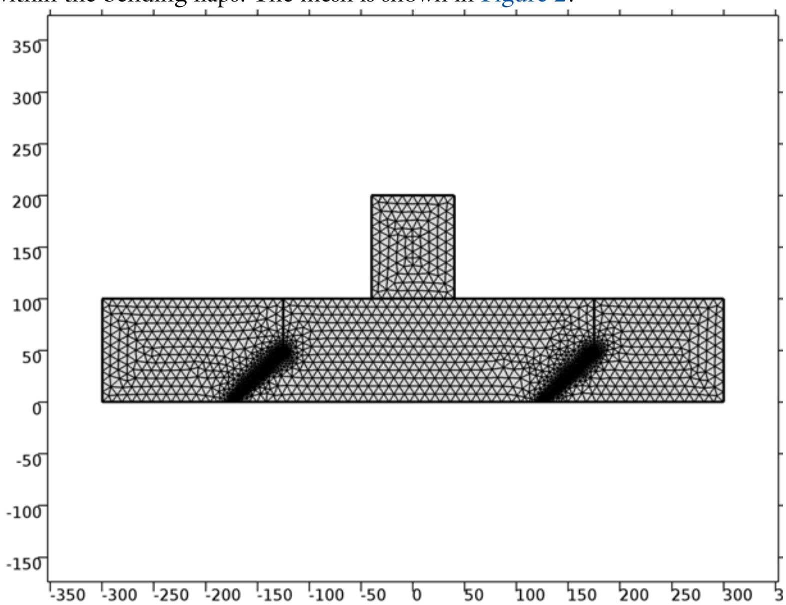


Figure 2: Initial mesh prior to any structural deformation.

A single time dependent study is performed over a duration of 2 seconds, which corresponds to two full oscillations of the inlet velocity. Some minor amendments to the default solver sequence, described in the step-by-step instructions, are required to ensure reliable results. In particular, the Time Stepping is set to `Strict` with the maximum step set to 0.01 s. This is needed because the feedback between the flap deformation and the fluid flow can lead to abrupt changes in the flow. The default time stepping setting allows the solver to vary the time step it takes so that larger steps can be taken during times when the solution is not rapidly varying. However, because of the potential for sudden changes to the fluid flow in fluid-structure interaction models it can be helpful to impose a maximum time step to ensure that rapid changes are not missed.

It should be noted that the geometry is parameterized so that the channel dimensions and flap angle can be straightforwardly modified by amending the relevant entry in the Global Parameters table. The average flow rate at the inlet and the fluid properties are also parameterized in the same way. In addition, the effective Reynolds number can be easily changed as the viscosity and average inlet velocity are appropriately scaled by a shared coefficient (the parameter `coeff`) that is computed from the target Reynolds number (the parameter `Re`). Note that the parameter `Re_check` is provided to confirm that the Reynolds number does indeed take the specified value. The model is setup with a Reynolds number of 16, but it is straightforward to verify that the pumping action occurs even for Reynolds numbers significantly less than one.

Results and Discussion

The mechanism by which the flow direction is regulated can be observed in the Flow and Stress default plot group. During the downstroke, when fluid is pushed from the vertical chamber into the channel, the right-hand flap is bent down towards the bottom of the channel whilst the left-hand flap is bend away from the channel bottom. This configuration is shown in Figure 3, which shows the solution at a time of 0.26 s which corresponds to when the velocity flowing into the vertical chamber via the inlet is at its maximum. Due to the asymmetric bending of the flaps, fluid can more easily flow out of the right-hand outlet. During the upstroke, when fluid is drawn from the channel into the vertical chamber, the flaps are bent in the opposite directions. This configuration is shown in Figure 4, which shows the solution at a time of 0.74 s. Now the right hand flap restricts the flow more than the left-hand flap, and the majority of the fluid that is drawn into the vertical chamber is from the left-hand outlet.

The result of this behavior is that there is a net flow rate from left-to-right inside the channel. This has many possible applications in microfluidic systems. For example, this device could be used to deliver fluid from a droplet reservoir connected to the left-hand outlet into a microfluidic pathway connected to the right-hand outlet. Alternately, this device could be used to create a circulating system where a fluid is pumped around a continuous loop to cool a microelectronic system.

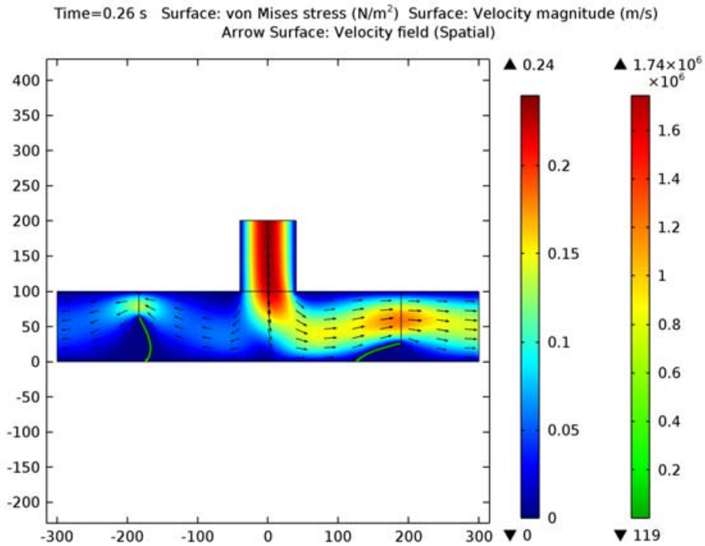


Figure 3: Velocity magnitude and velocity field, along with the von Mises stress within the flaps, during the pumping downstroke.

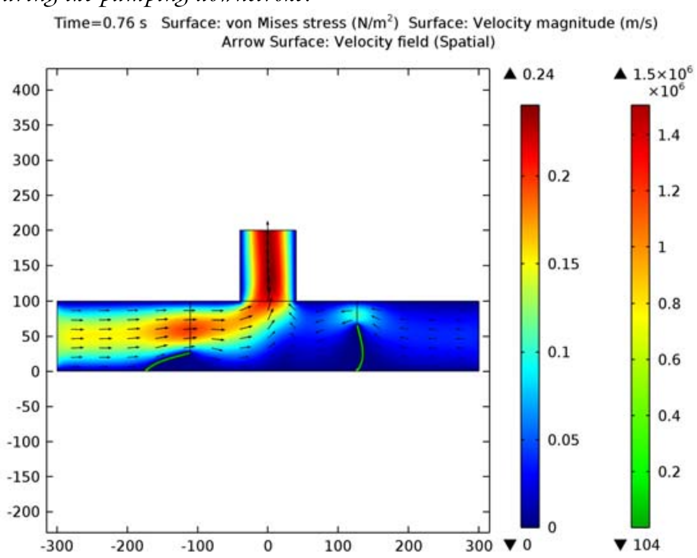


Figure 4: Velocity magnitude and velocity field, along with von Mises stress within the flaps, during the pumping upstroke.

The net volume of fluid that is pumped from left-to-right is shown in Figure 5. As expected, the gradient of the curve, which is the net flow rate, varies sinusoidally with a period equal to the inlet velocity. The maximum gradients occur at intervals of odd multiples of 0.5 s, which correspond with the peaks in the magnitude of the inlet velocity.

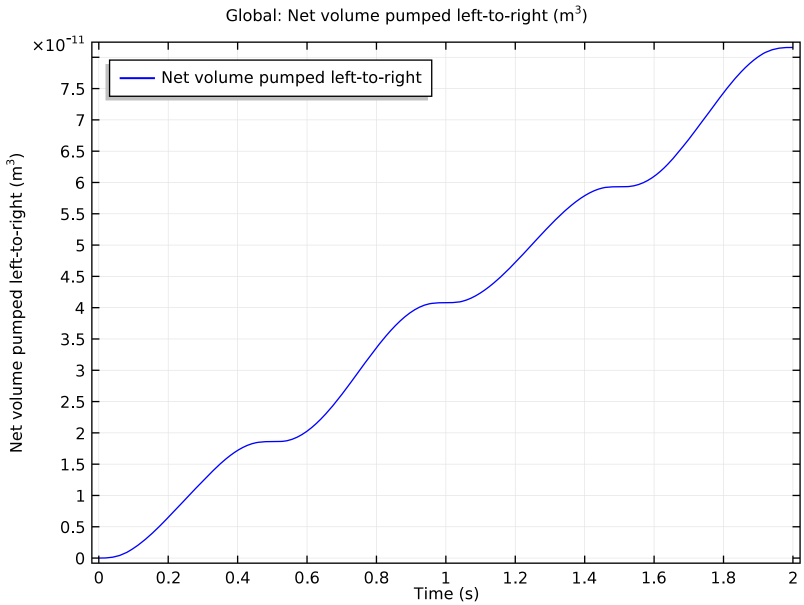


Figure 5: Net volume pumped from left-to-right as a function of time.

Application Library path: `MEMS_Module/Fluid-Structure_Interaction/micropump_mechanism`

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 **2D**.
2 In the **Select Physics** tree, select **Fluid Flow>Fluid-Structure Interaction (fsi)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6 Click **Done**.

Add some parameters which will be used to control the fluid properties and device geometry.

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:

Name	**Expression**	**Value**	**Description**
Re	16	16	Reynolds number
coeff	4/sqrt(Re)	1	Coefficient to change Reynolds number
dens	1000[kg/m^3]	1000 kg/m³	Fluid density
visc	0.001[Pa*s]*coeff	0.001 Pa·s	Fluid dynamic viscosity
U	16[cm/s]/coeff	0.16 m/s	Average inlet flow speed
H	100[um]	1E-4 m	Channel height
W	10[um]	1E-5 m	Domain width
rp	2[um]	2E-6 m	Pillar radius
hp	70[um]	7E-5 m	Pillar height
L	600[um]	6E-4 m	Length of channel
beta	45[deg]	0.7854 rad	Flap tilt angle
x0	150[um]	1.5E-4 m	Flap center location
Re_check	dens*U*H/(visc)	16	Reynolds number

Next create the geometry for the device. Notice how vertical lines are used to connect the tip of each rod to the top of the channel. This is not needed to create the desired geometry, however these lines are very helpful for limiting the deformation of the mesh elements as the rods bend. It is good practice to include boundaries to guide the deformation of a moving mesh.

GEOMETRY 1

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**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

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

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

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

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

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

4 In the **Height** text field, type `2*hp`.

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

6 In the **x** text field, type `x0-hp*sin(beta)/2`.

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

Copy 1 (copy1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **r1** only.

Intersection 1 (int1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

2 Select the objects **r1** and **r2** only.

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

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

Fillet 1 (fil1)

1 On the **Geometry** toolbar, click **Fillet**.

2 On the object **int1**, select Points 3 and 4 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `rp`.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **x** to `x0+hp*sin(beta)/2` and **y** to `hp*cos(beta)`.

5 In row **2**, set **x** to `x0+hp*sin(beta)/2` and **y** to `H`.

Copy 2 (copy2)

1 On the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 Select the objects **b1** and **fil1** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **x** text field, type `-2*x0`.

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.8*H`.

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

5 Locate the **Position** section. In the **x** text field, type `-0.4*H`.

6 In the **y** text field, type `H`.

7 Click **Build All Objects**.

Add two **Integration Component Coupling** operators, one to the boundary which forms the left outlet and one to the boundary which forms the right outlet. These operators will be used to calculate the flow rate out of each outlet.

DEFINITIONS

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intopL` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 1 only.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intopR` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 21 only.

Add two more **Integration Component Coupling** operators. These integration operators are used to calculate the change in position of the tip of the rods as they bend, so that this information can be used to help control the mesh as it adapts to the changing geometry. These will be used when setting the prescribed mesh displacement settings in the **Fluid-Structure Interaction** physics interface. In general, it is good practice to set up guide boundaries to help control how the mesh deforms. This model includes such boundaries as an example.

Integration 3 (intop3)

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 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `7` in the **Selection** text field.

6 Click **OK**.

Integration 4 (intop4)

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 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `17` in the **Selection** text field.

6 Click **OK**.

Create some variables to calculate the net flow rate in the channel. The first two variables calculate the average flow rate out of each outlet by performing an average over `u_fluid`, which is the horizontal component of the fluid flow. Note the sign convention, that assigns positive values to left-right flow and negative values to right-left flow. The third variable calculates the difference between the average flow rate out of each outlet, positive values correspond to a net flow from left to right.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
UoutL	-intopL(u_fluid)*W	m³/s	Flow rate from left outlet
UoutR	intopR(u_fluid)*W	m³/s	Flow rate from right outlet
UoutNet	UoutR-UoutL	m³/s	Net flow rate

In order to integrate the average net flow rate over time to calculate the volume of fluid that is pumped, add a Global ODE and DAEs physics interface to the model.

ADD PHYSICS

1. On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2. Go to the **Add Physics** window.
3. In the tree, select **Mathematics>ODE and DAE Interfaces>Global ODEs and DAEs (ge)**.
4. Click **Add to Component** in the window toolbar.

GLOBAL ODES AND DAES (GE)

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

Global Equations 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Global ODEs and DAEs (ge)** click **Global Equations 1**.
2. In the **Settings** window for **Global Equations**, locate the **Global Equations** section.
3. In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
Vpump	Vpumpt-UoutNet	0	0	Net volume pumped left-to-right

4. Locate the **Units** section. Find the **Dependent variable quantity** subsection. From the list, choose **None**.
5. In the **Unit** text field, type `m^3`.
6. Find the **Source term quantity** subsection. From the list, choose **None**.
7. In the **Unit** text field, type `m^3/s`.

   Configure the **Fluid-Structure Interaction** physics interface.

FLUID-STRUCTURE INTERACTION (FSI)

On the **Physics** toolbar, click **Global ODEs and DAEs (ge)** and choose **Fluid-Structure Interaction (fsi)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Fluid-Structure Interaction (fsi)**.

2 In the **Settings** window for **Fluid-Structure Interaction**, locate the **Free Deformation Settings** section.

3 From the **Mesh smoothing type** list, choose **Yeoh**.

4 Locate the **Physical Model** section. From the **Compressibility** list, choose **Incompressible flow**.

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Fluid-Structure Interaction (fsi)** click **Linear Elastic Material 1**.

2 Select Domains 2 and 5 only.

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

4 Select the **Nearly incompressible material** check box.

Inlet 1

1 In the **Model Builder** window, right-click **Fluid-Structure Interaction (fsi)** and choose the boundary condition **Laminar Flow>Inlet**.

2 Select Boundary 12 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `U*6*s*(1-s)*sin(2*pi*t/(1[s]))`.

Outlet 1

1 Right-click **Fluid-Structure Interaction (fsi)** and choose the boundary condition **Laminar Flow>Outlet**.

2 Select Boundaries 1 and 21 only.

3 In the **Settings** window for **Outlet**, locate the **Pressure Conditions** section.

4 Clear the **Suppress backflow** check box.

Fixed Constraint 1

1 Right-click **Fluid-Structure Interaction (fsi)** and choose the boundary condition **Solid Mechanics>Fixed Constraint**.

2 Select Boundaries 4 and 15 only.

Prescribed Mesh Displacement 2

1 Right-click **Fluid-Structure Interaction (fsi)** and choose **Prescribed Mesh Displacement**.

2 Select Boundaries 3, 9, 14, and 20 only.

3 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Prescribed Mesh Displacement** section.

4 Clear the **Prescribed x displacement** check box.

Prescribed Mesh Displacement 3

1 Right-click **Fluid-Structure Interaction (fsi)** and choose **Prescribed Mesh Displacement**.

2 Select Boundary 8 only.

3 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Prescribed Mesh Displacement** section.

4 Clear the **Prescribed y displacement** check box.

5 In the d_x text field, type `intop3(x-X)`.

Prescribed Mesh Displacement 4

1 Right-click **Fluid-Structure Interaction (fsi)** and choose **Prescribed Mesh Displacement**.

2 Select Boundary 19 only.

3 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Prescribed Mesh Displacement** section.

4 Clear the **Prescribed y displacement** check box.

5 In the d_x text field, type `intop4(x-X)`.

Prescribed Deformation 1

1 Right-click **Fluid-Structure Interaction (fsi)** and choose **Prescribed Deformation**.

2 Select Domain 4 only.

Prescribed Mesh Displacement 5

1 Right-click **Fluid-Structure Interaction (fsi)** and choose **Prescribed Mesh Displacement**.

2 Select Boundary 11 only.

With the physics interfaces configured, add materials to the geometry domains. In this case, it was beneficial to wait until all the physics settings were configured before adding materials, as COMSOL automatically adjusts which materials properties are needed depending on the physics assigned to each domain.

MATERIALS

Material 1 (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, type `Fluid` in the **Label** text field.
3. Select Domains 1, 3, 4, and 6 only.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Density	rho	`dens`	kg/m³	Basic
Dynamic viscosity	mu	`visc`	Pa·s	Basic

Material 2 (mat2)

1. Right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, type `Solid` in the **Label** text field.
3. Select Domains 2 and 5 only.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	`3.6e5`	Pa	Basic
Poisson's ratio	nu	`0.499`	1	Basic
Density	rho	`970`	kg/m³	Basic

Configure the mesh.

MESH 1

Free Triangular 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.
2. In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Entire geometry**.

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 `10`.

5 In the **Minimum element size** text field, type `1.4`.

6 In the **Maximum element growth rate** text field, type `1.4`.

7 In the **Curvature factor** text field, type `0.6`.

8 In the **Resolution of narrow regions** text field, type `0.7`.

Size 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.

2 Right-click **Size 1** and choose **Move Up**.

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

4 From the **Geometric entity level** list, choose **Domain**.

5 Select Domains 2 and 5 only.

6 Locate the **Element Size** section. Click the **Custom** button.

7 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

8 In the associated text field, type `2`.

9 Select the **Minimum element size** check box.

10 In the associated text field, type `1.5`.

11 In the **Model Builder** window, click **Mesh 1**.

12 In the **Settings** window for **Mesh**, click **Build All**.

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

Configure the study, some minor changes to the default solver configuration are needed.

STUDY 1

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

Set the time steps and range for the study. Also tighten the time-dependent solver tolerance to make the time steps more conservative.

Solution 1 (sol1)

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Times** text field, type `range(0,0.02,2)`.

3 From the **Tolerance** list, choose **User controlled**.

4 In the **Relative tolerance** text field, type `0.001`.

Alter the time stepping in the **Time-Dependent Solver 1** node. By default, the time-dependent solver varies the time step automatically as it solves. However, for FSI simulations it is often useful to constrain this by setting a maximum step size. This improves accuracy by ensuring that abrupt changes in flow are not missed.

5 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

6 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

7 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Intermediate**.

8 Select the **Maximum step** check box.

9 In the associated text field, type `0.005`.

The **Fully Coupled** solver is often most appropriate for FSI simulations. A **Fully Coupled** node can be added to the **Time-Dependent Solver 1** sequence, when this is done the default **Segregated** solver is overridden.

10 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** node.

11 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** and choose **Fully Coupled**.

12 In the **Settings** window for **Fully Coupled**, click to expand the **Method and termination** section.

13 Locate the **Method and Termination** section. In the **Maximum number of iterations** text field, type `6`.

14 From the **Jacobian update** list, choose **Once per time step**.

Now the study can be solved.

15 On the **Study** toolbar, click **Compute**.

The default **Flow and Stress** plot group shows a color map of the von Mises stress within the flexible rods, as well as the velocity magnitude of the fluid within the channel. An **Arrow Surface** plot is of the fluid velocity is superimposed over the colormap. An animation of the data series allows the action of the pump to be visualized.

RESULTS

Animation 1

1. On the **Results** toolbar, click **Animation** and choose **File**.
2. In the **Settings** window for **Animation**, locate the **Target** section.
3. From the **Target** list, choose **Player**.
4. Locate the **Frames** section. From the **Frame selection** list, choose **All**.
5. Right-click **Animation 1** and choose **Play**.

   The animation demonstrates how the passive motion of the rods as they react to the fluid results in a net flow of fluid from left-to-right. During the "downstoke", when fluid flows from the inlet down into the channel, the right rod bends towards the bottom of the channel whilst the left rod bend away from the bottom. This restricts the flow towards the left-hand outlet, relative to the right-hand outlet. During the "upstroke", where fluid flows from the channel up towards the inlet, the rods bend in the opposite directions. Now the inward flow from the right-hand outlet towards the center of the channel is restricted. This results in a net flow from left-to-right, where fluid is drawn in from the left-hand outlet and pushed out of the right-hand outlet.

   The 1D plot group, which plots `Vpump` from the **Global ODEs and DAEs** interface, confirms that there is indeed a net flow from left-to-right as expected.

1D Plot Group 3

1. In the **Model Builder** window, under **Results** click **1D Plot Group 3**.
2. In the **Settings** window for **1D Plot Group**, type `Net volume pumped left-to-right` in the **Label** text field.
3. Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

Microresistor Beam

Introduction

This example illustrates the ability to couple thermal, electrical, and structural analysis in one model. This particular application moves a beam by passing a current through it; the current generates heat, and the temperature increase leads to displacement through thermal expansion. The model estimates how much current and increase in temperature are necessary to displace the beam.

Although the model involves a rather simple 3D geometry and straightforward physics, it provides a good example of multiphysics modeling.

Model Definition

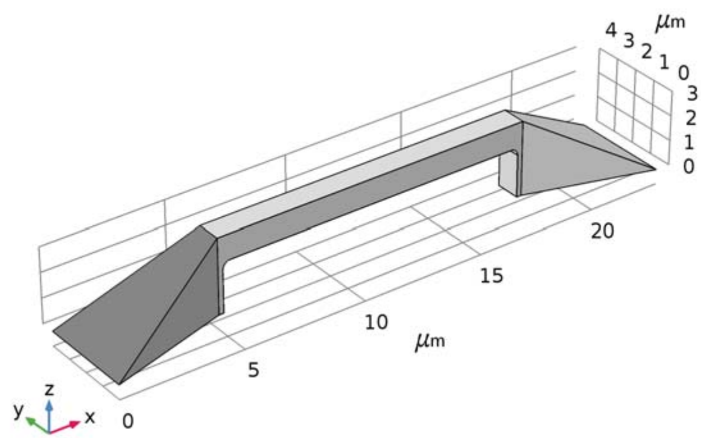


Figure 1: Microbeam geometry.

A copper microbeam has a length of 13 µm with a height and width of 1 µm. Feet at both ends bond it rigidly to a substrate. An electric potential of 0.2 V applied between the feet induces an electric current. Due to the material's resistivity, the current heats up the structure. Because the beam operates in the open, the generated heat dissipates into the air. The thermally induced stress loads the material and deforms the beam.

As a first approximation, you can assume that the electrical conductivity is constant. However, a conductor's resistivity increases with temperature. In the case of copper, the relationship between resistivity and temperature is approximately linear over a wide range of temperatures:

$$\rho = \rho_0(1 + \alpha(T - T_0)) \tag{1}$$

α is the temperature coefficient. You obtain the conductor's temperature dependency from the relationship that defines electric resistivity; conductivity is simply its reciprocal ($\sigma = 1/\rho$).

For the heat transfer equations, set the base boundaries facing the substrate to a constant temperature of 323 K. You model the convective air cooling in other boundaries using a heat flux boundary condition with a heat transfer coefficient, h, of 5 W/(m^2·K) and an external temperature, T_{inf}, of 298 K. Standard constraints handle the bases' rigid connection to the substrate.

Results and Discussion

Figure 2 shows the temperature field on the microbeam surface when solving the model using a temperature-dependent resistivity as in Equation 1. Based on the color scale, the maximum temperature is about 710 K.

Figure 3 shows the microbeam's deformation. The displacement for the temperature-dependent case is 48 nm compared to the maximum displacement for constant electrical conductivity, which is 88 nm (the plot scales the deformation by a factor of around 20).

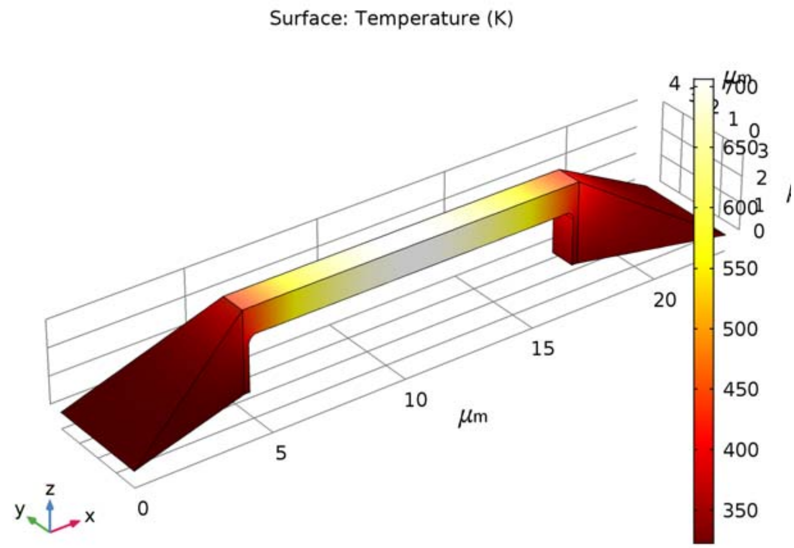


Figure 2: Surface temperature with temperature-dependent electrical conductivity.

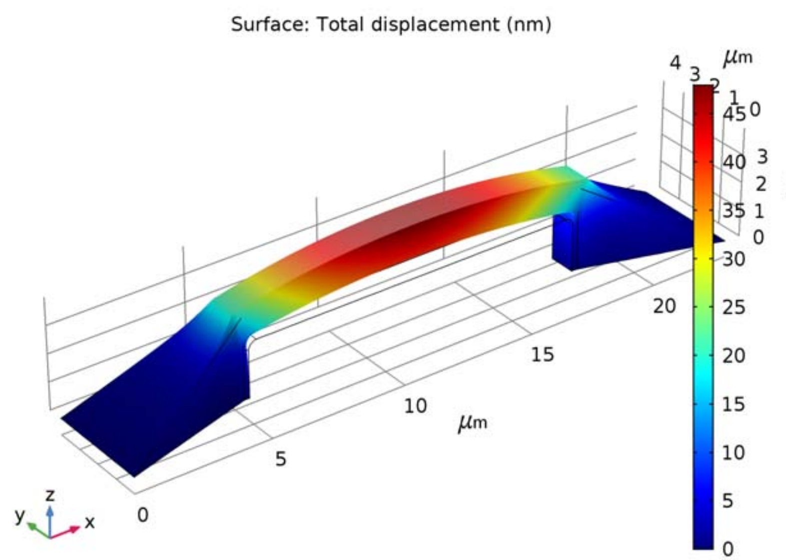


Figure 3: Microbeam deformation with temperature-dependent electrical conductivity.

Notes About the COMSOL Implementation

In this example you create the 3D geometry by starting with two 2D work planes. The first one views the geometry from above, and the second does so from the side. You create cross sections on the work planes, which you then extrude into 3D. As the final step you create the resistor beam geometry as the intersection of the extruded objects. You can also skip the step-by-step instructions for the geometry creation and import the ready-made geometry directly from the Application Libraries.

By using the *Joule Heating and Thermal Expansion* predefined multiphysics interface you automatically add the equations for three physics including the necessary multiphysics couplings. In this case the physics equations describe the current and heat conduction and structural mechanics problems. The interface also provides suitable defaults for the solver.

Application Library path: `MEMS_Module/Actuators/microresistor_beam`

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>Joule Heating and Thermal Expansion**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
6. Click **Done**.

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:

Name	Expression	Value	Description
V0	0.2[V]	0.2 V	Applied voltage
T0	323[K]	323 K	Heat sink temperature
Text	298[K]	298 K	External temperature
k	5[W/(m^2*K)]	5 W/(m²·K)	Heat transfer coefficient

GEOMETRY 1

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**.

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Bézier Polygon 1 (b1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **xw** to 5 and **yw** to 1.5.

5 Find the **Added segments** subsection. Click **Add Linear**.

6 Find the **Control points** subsection. In row **2**, set **xw** to 18.

7 Find the **Added segments** subsection. Click **Add Linear**.

8 Find the **Control points** subsection. In row **2**, set **xw** to 23 and **yw** to 0.

9 Find the **Added segments** subsection. Click **Add Linear**.

10 Find the **Control points** subsection. In row **2**, set **yw** to 4.

11 Find the **Added segments** subsection. Click **Add Linear**.

12 Find the **Control points** subsection. In row **2**, set **xw** to 18 and **yw** to 2.5.

13 Find the **Added segments** subsection. Click **Add Linear**.

14 Find the **Control points** subsection. In row **2**, set **xw** to 5.

15 Find the **Added segments** subsection. Click **Add Linear**.

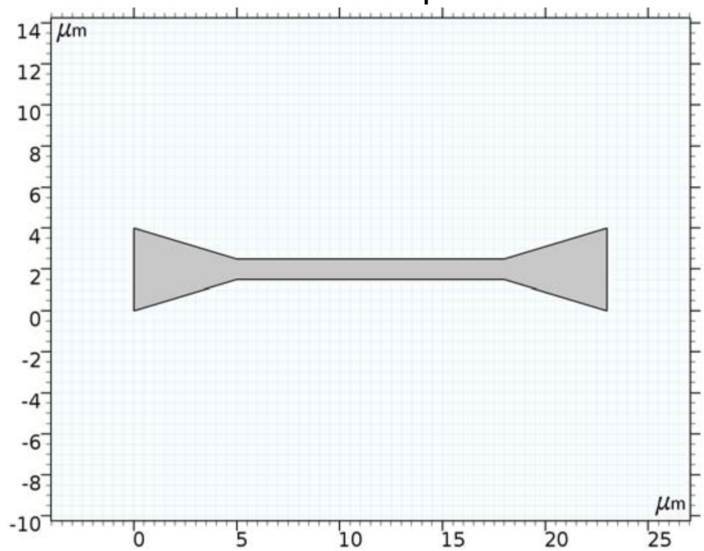
16 Find the **Control points** subsection. In row **2**, set **xw** to 0 and **yw** to 4.

17 Find the **Added segments** subsection. Click **Add Linear**.

18 Find the **Control points** subsection. Click **Close Curve**.

19 On the **Work Plane** toolbar, click **Build All**.

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



Work Plane 1 (wp1)

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

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

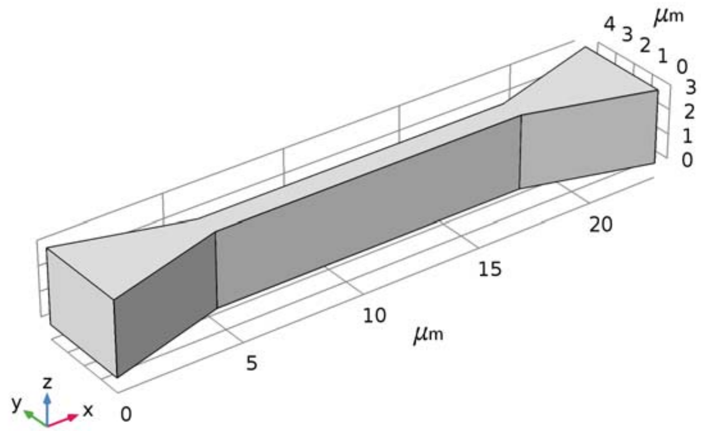
2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (µm)
3

4 Click **Build All Objects**.

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



Work Plane 2 (wp2)

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

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

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

4 Find the **Planar face** subsection. Select the **Active** toggle button.

5 On the object **ext1**, select Boundary 6 only.

6 In the **Offset in normal direction** text field, type `-1.5`.

7 Select the **Reverse normal direction** check box.

8 Click **Show Work Plane**.

Plane Geometry

1 In the **Settings** window for **Plane Geometry**, locate the **Visualization** section.

2 Find the **In-plane visualization of 3D geometry** subsection. Clear the **Intersection (cyan)** check box.

3 Clear the **Coincident entities (blue)** check box.

Bézier Polygon 1 (b1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **xw** to `-11.5` and **yw** to `-1.5`.

5 In row **2**, set **xw** to `-6.3` and **yw** to `-1.5`.

6 Find the **Added segments** subsection. Click **Add Linear**.

7 Find the **Control points** subsection. In row **2**, set **yw** to `0.5`.

8 Find the **Added segments** subsection. Click **Add Linear**.

9 Find the **Control points** subsection. In row **2**, set **xw** to `6.3`.

10 Find the **Added segments** subsection. Click **Add Linear**.

11 Find the **Control points** subsection. In row **2**, set **yw** to `-1.5`.

12 Find the **Added segments** subsection. Click **Add Linear**.

13 Find the **Control points** subsection. In row **2**, set **xw** to `11.5`.

14 Find the **Added segments** subsection. Click **Add Linear**.

15 Find the **Control points** subsection. In row **2**, set **xw** to `6.5` and **yw** to `1.5`.

16 Find the **Added segments** subsection. Click **Add Linear**.

17 Find the **Control points** subsection. In row **2**, set **xw** to `-6.5`.

18 Find the **Added segments** subsection. Click **Add Linear**.

19 Find the **Control points** subsection. Click **Close Curve**.

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

Fillet 1 (fil1)

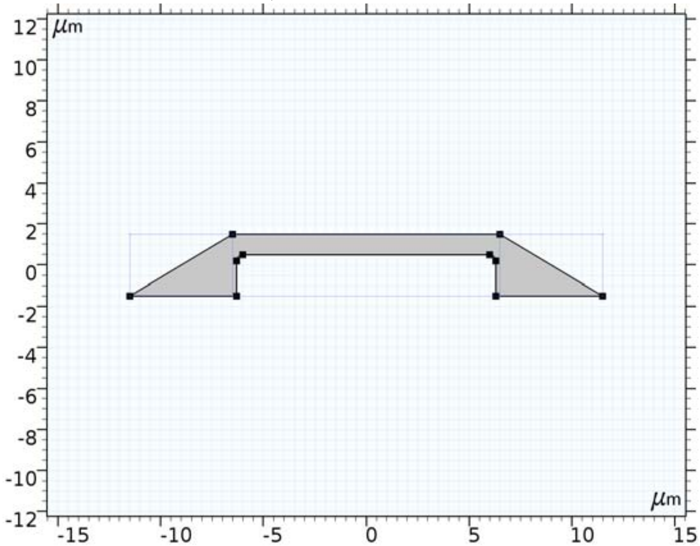
1 On the **Work Plane** toolbar, click **Fillet**.

2 On the object **b1**, select Points 4 and 6 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

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

5 On the **Work Plane** toolbar, click **Build All**.



Work Plane 2 (wp2)

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

Extrude 2 (ext2)

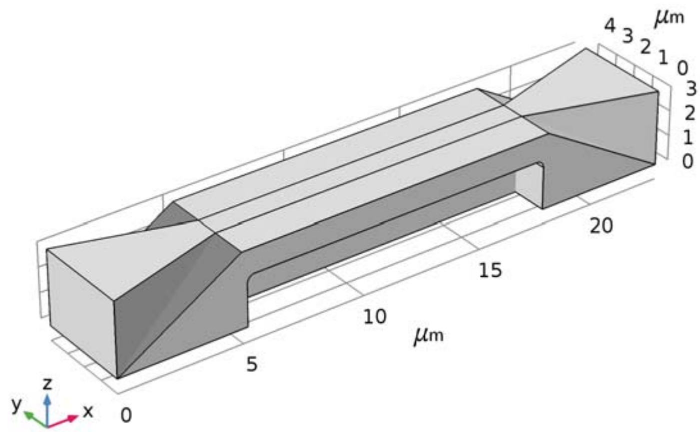
1 On the **Geometry** toolbar, click **Extrude**.

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

3 In the table, enter the following settings:

Distances (µm)
4

4 Click **Build All Objects**.



Intersection 1 (int1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

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

3 In the **Settings** window for **Intersection**, click **Build All Objects**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

The model geometry is now complete.

DEFINITIONS

Add a set of selections that you can use later when applying boundary conditions.

Explicit 1

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

2 In the **Model Builder** window, right-click **Explicit 1** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `connector1` in the **New label** text field.

4 Click **OK**.

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

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

7 Select Boundary 1 only.

Explicit 2

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

2 In the **Model Builder** window, right-click **Explicit 2** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `connector2` in the **New label** text field.

4 Click **OK**.

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

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

7 Select Boundary 13 only.

Explicit 3

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

2 In the **Model Builder** window, right-click **Explicit 3** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `connectors` in the **New label** text field.

4 Click **OK**.

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

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

7 Select Boundaries 1 and 13 only.

ADD MATERIAL

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

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

3 In the tree, select **MEMS>Metals>Cu - Copper**.

4 Click **Add to Component** in the window toolbar.

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

MATERIALS

Cu - Copper (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Cu - Copper (mat1)**.

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	1	1	Basic

4 Click to expand the **Material properties** section. Locate the **Material Properties** section. In the **Material properties** tree, select **Electromagnetic Models>Linearized Resistivity>Reference resistivity (rho0)**.

5 Click **Add to Material**.

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

Property	Name	Value	Unit	Property group
Reference resistivity	rho0	`1.72e-8[ohm*m]`	Ω·m	Linearized resistivity
Resistivity temperature coefficient	alpha	`0.0039[1/K]`	1/K	Linearized resistivity
Reference temperature	Tref	`293[K]`	K	Linearized resistivity

ELECTRIC CURRENTS (EC)

Current Conservation 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electric Currents (ec)** click **Current Conservation 1**.

2 In the **Settings** window for **Current Conservation**, locate the **Conduction Current** section.

3 From the σ list, choose **Linearized resistivity**.

Before solving the two-way coupled model with a temperature-dependent resistivity, use a constant resistivity for later comparison:

4 From the α list, choose **User defined**. Keep the default zero value for α.

Ground 1

1 In the **Model Builder** window, right-click **Electric Currents (ec)** and choose **Ground**.

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

3 From the **Selection** list, choose **connector2**.

Electric Potential 1

1 Right-click **Electric Currents (ec)** and choose **Electric Potential**.

2 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

3 In the V_0 text field, type `V0`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **connector1**.

MULTIPHYSICS

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Thermal Expansion 1 (te1)**.

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

3 In the T_{ref} text field, type `Text`.

HEAT TRANSFER IN SOLIDS (HT)

On the **Physics** toolbar, click **Electric Currents (ec)** and choose **Heat Transfer in Solids (ht)**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, type `T0` in the T text field.

Heat Flux 1

1 In the **Model Builder** window, right-click **Heat Transfer in Solids (ht)** and choose **Heat Flux**.

2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Select all boundaries for simplicity; next you will add a node that overrides this boundary condition for the connectors.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the h text field, type `k`.

6 In the T_{ext} text field, type `Text`.

Temperature 1

1 Right-click **Heat Transfer in Solids (ht)** and choose **Temperature**.

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

3 In the T_0 text field, type `T0`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **connectors**.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.
2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **connectors**.

MESH 1

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Predefined** list, choose **Finer**.
4 In the **Model Builder** window, click **Mesh 1**.
5 In the **Settings** window for **Mesh**, click **Build All**.

STUDY 1

You can use the default solver settings for this model.

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

RESULTS

Stress (solid)

The first default plot presents a surface plot of the von Mises stress. Modify it to show the displacement magnitude.

1 In the **Model Builder** window, right-click **Stress (solid)** and choose **Rename**.
2 In the **Rename 3D Plot Group** dialog box, type `Displacement - Study 1` in the **New label** text field.
3 Click **OK**.

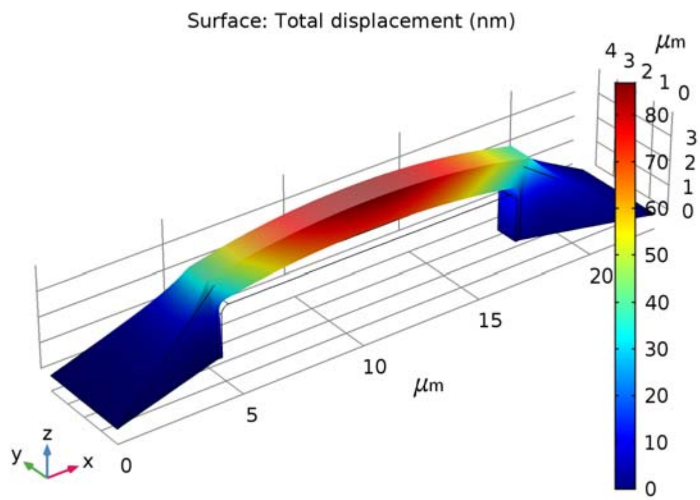
Surface 1

1 In the **Model Builder** window, expand the **Results>Displacement - Study 1** node, then click **Surface 1**.
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>Solid Mechanics> Displacement>solid.disp - Total displacement**.

3 Locate the **Expression** section. From the **Unit** list, choose **nm**.

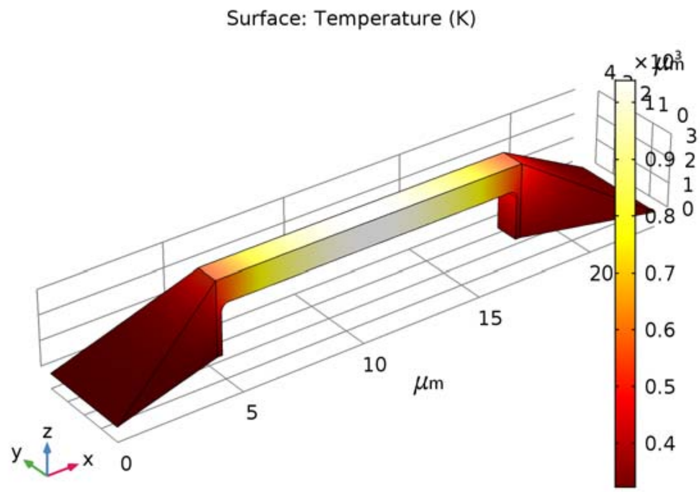
4 On the **Displacement - Study 1** toolbar, click **Plot**.

As the color legend shows, the maximum displacement is roughly 88 nm with a constant resistivity.



Temperature (ht)

The second default surface plot shows the temperature field. Note the maximum temperature of roughly 1048 K.



Now restore the temperature-dependence of the resistivity that you temporarily disabled and then add a new study and solve the model again.

ELECTRIC CURRENTS (EC)

On the **Physics** toolbar, click **Solid Mechanics (solid)** and choose **Electric Currents (ec)**.

Current Conservation 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Electric Currents (ec)** click **Current Conservation 1**.
2. In the **Settings** window for **Current Conservation**, locate the **Conduction Current** section.
3. From the α list, choose **From material**.

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

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

RESULTS

Temperature (ht) 1

As you can see from the plot, using the more realistic material model with a temperature-dependent resistivity has a significant effect on the solution. The maximum temperature is now almost 340 K lower.

Stress (solid)

1 In the **Model Builder** window, under **Results** right-click **Stress (solid)** and choose **Rename**.
2 In the **Rename 3D Plot Group** dialog box, type `Displacement - Study 2` in the **New label** text field.
3 Click **OK**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Displacement - Study 2** node, then click **Surface 1**.
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>Solid Mechanics> Displacement>solid.disp - Total displacement**.
3 Locate the **Expression** section. From the **Unit** list, choose **nm**.
4 On the **Displacement - Study 2** toolbar, click **Plot**.

   Similarly, the maximum displacement has been reduced from 88 nm to around 50 nm.

Created in COMSOL Multiphysics 5.3

Piezoceramic Tube

Introduction

This example involves a static 2D axisymmetric analysis of a piezoelectric actuator using the Piezoelectric Devices multiphysics interface. It models a radially polarized piezoelectric tube, as described by S. Peelamedu and co-authors (Ref. 1). An application area where radially polarized tubes are employed is in nozzles for fluid control in inkjet printers.

Model Definition

GEOMETRY

The tube has a height of 0.62 mm and an inner and outer radius of 0.38 mm and 0.62 mm, respectively. It is represented in an axisymmetric geometry by a single off-axis rectangle, as shown in Figure 1.

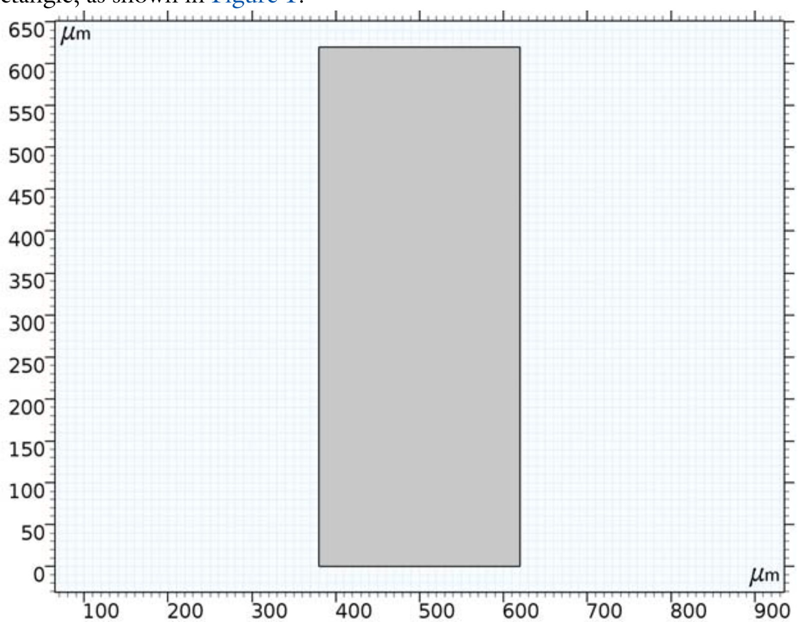


Figure 1: The axisymmetric geometry. Length units on x- and y-axes are shown in μm.

BOUNDARY CONDITIONS

The model studies two cases distinguished by different boundary conditions. Case 1 represents the direct piezoelectric effect, and Case 2 represents the inverse piezoelectric effect.

Case 1—Direct Piezoelectric Effect:

- Structural mechanics boundary condition—constrain the bottom surface from moving axially (in the z direction), but also add an internal fluid pressure of 0.1 MPa.
- Electrostatics boundary condition—ground the inner and outer surfaces.

Case 2—Inverse Piezoelectric Effect:

- Structural mechanics boundary condition—constrain the bottom surface from moving axially (in the z-direction).
- Electrostatics boundary condition—apply a 1 V potential difference between the tube's inner and outer surfaces.

MATERIAL ORIENTATION

COMSOL's material library data is entered in a form which assumes that the crystal polarization is aligned with the global co-ordinate z axis. For the radially polarized case treated in this model, the orientation must be rotated so that the material polarization direction is aligned with the r direction (radially polarized). To do so, specify the co-ordinate system in the Piezoelectric Material feature. By selecting the co-ordinate system as the predefined zx-plane system, you rotate the material so that its z direction is aligned with the r direction of the model, and the material's x direction is aligned with the model's z direction.

The piezoceramic material in this example (PZT-5H) is a transversely isotropic material, which is a special class of orthotropic materials. Such a material has the same properties in one plane (isotropic behavior) and different properties in the direction normal to this plane. Thus you can use either the zx-plane material orientation or the zy-plane material orientation; both give the same solution.

Results and Discussion

Figure 2 shows the radial displacement due to the applied pressure in Case 1, and Figure 3 shows the corresponding induced electric potential. Both the radial displacement and potential are shown along a cut line 300 μm above the base of the tube in Figure 4 and Figure 5, receptively.

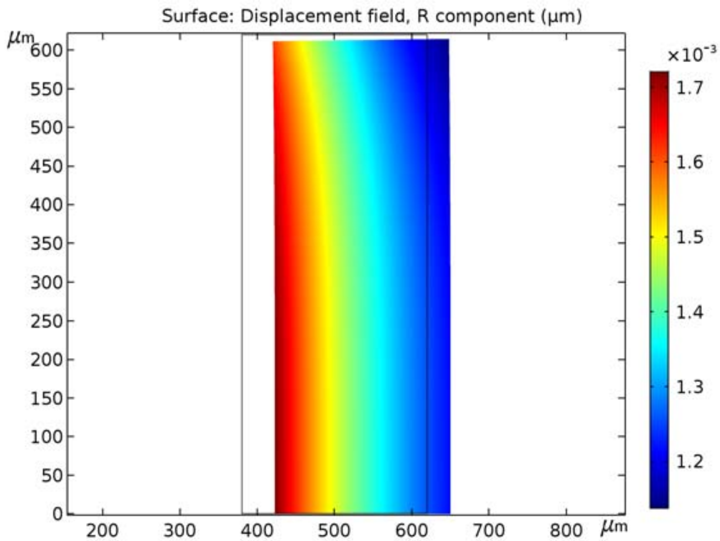


Figure 2: Deformed shape and radial displacement due to an internal pressure of 0.1 MPa (case 1 —the direct piezoelectric effect).

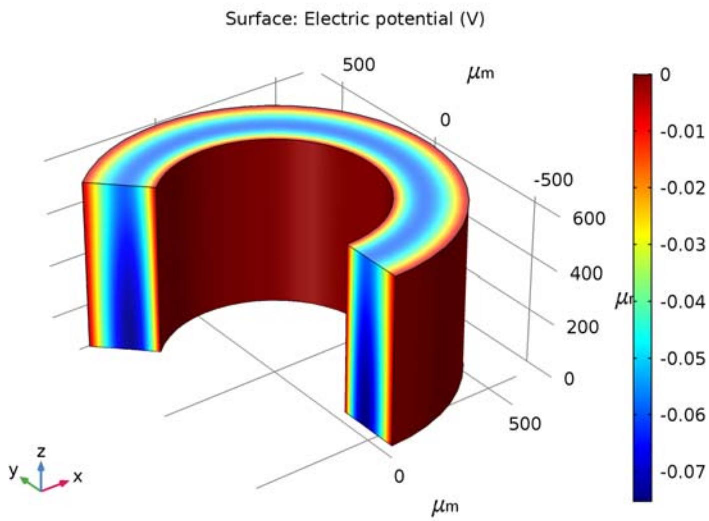


Figure 3: Induced electric potential within the deformed tube due to an internal pressure of 0.1 MPa (case 1 —the direct piezoelectric effect).

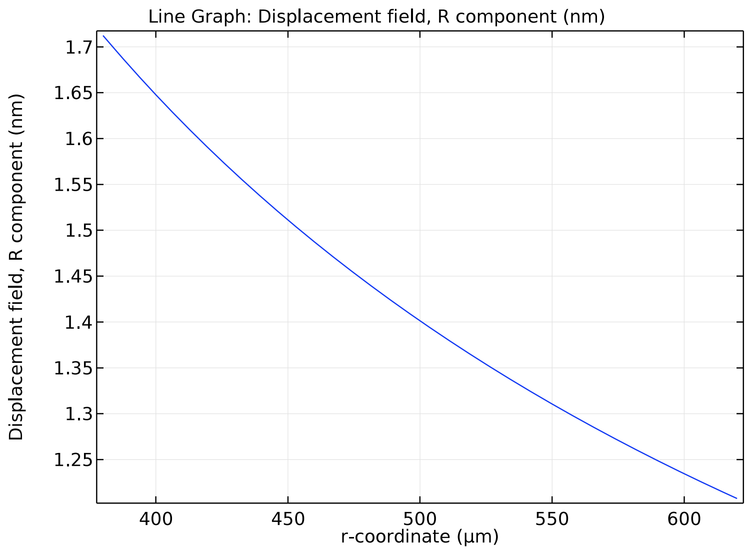


Figure 4: Radial displacement as a function of r-coordinate at a height of 300 µm above the base of the tube. The results are for Case 1—the direct piezoelectric effect.

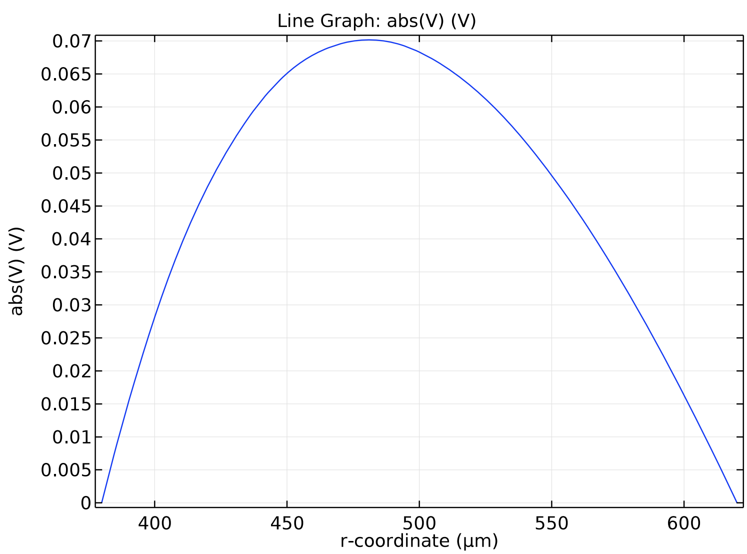


Figure 5: Electric potential as a function of r-coordinate at a height of 300 µm above the base of the tube. The results are for Case 1—the direct piezoelectric effect.

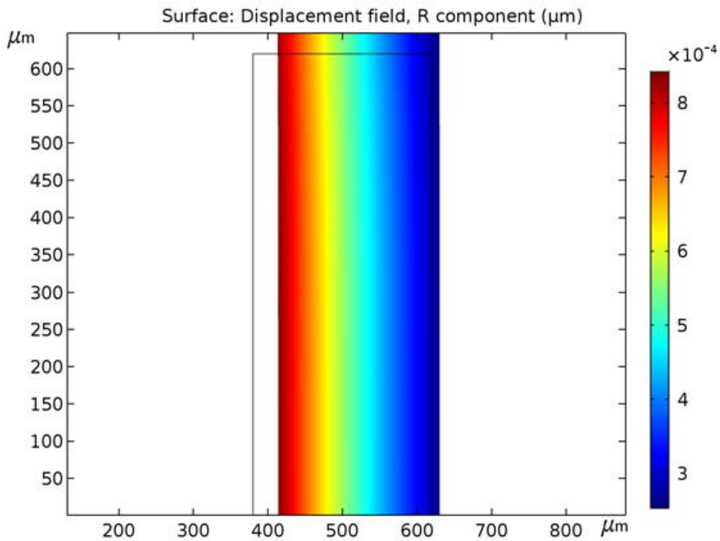


Figure 6: Deformed shape and radial displacement of the piezoceramic-tube actuator due to the radial electric field (Case 2—Inverse Piezoelectric Effect).

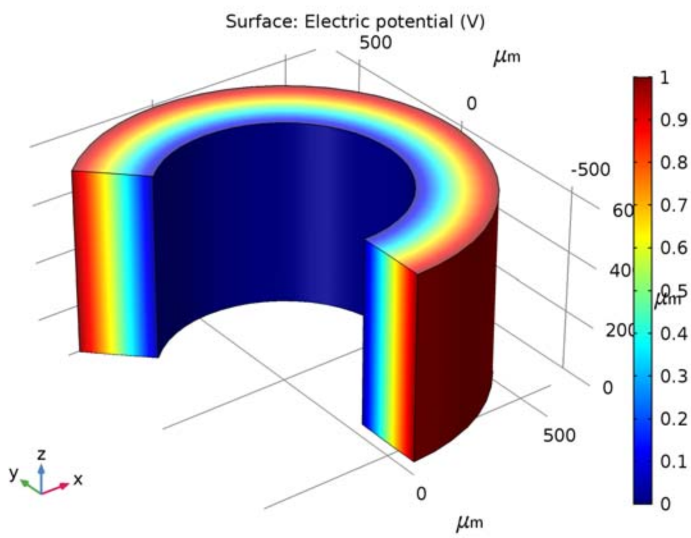


Figure 7: Electric potential applied to the tube to induce the displacements shown in Figure 6 *(Case 2—Inverse Piezoelectric Effect).*

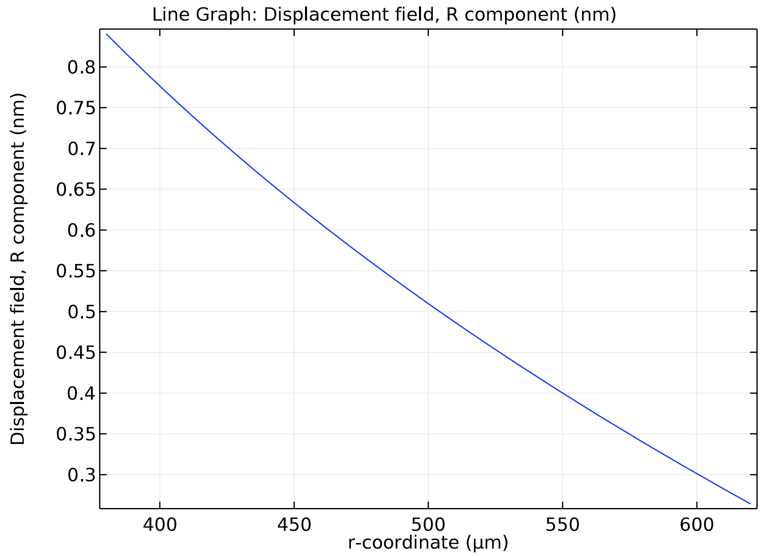


Figure 8: Radial displacement as a function of r-coordinate at a height of 300 µm above the base of the tube. The results are for Case 2—the inverse piezoelectric effect.

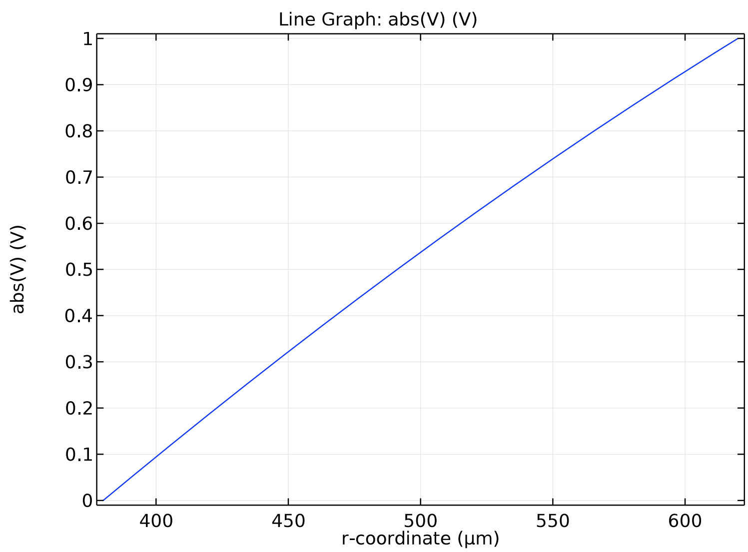


Figure 9: Electric potential as a function of r-coordinate at a height of 300 µm above the base of the tube. The results are for Case 2—the inverse piezoelectric effect.

Figure 6 shows the radial displacement resulting from the applied potential shown in Figure 7. The radial displacement and potential are shown along a cut line 300 μm above the base of the tube in Figure 8 and Figure 9, respectively.

These results show good agreement with those from S. Peelamedu (Ref. 1).

Reference

1. S.M. Peelamedu, C.B. Kosaraju, R.V. Dukkipati and N.G. Naganathan, "Numerical Approach for Axisymmetric Piezoceramic Geometries towards Fluid Control Applications, " *Proceedings of the Institution of Mechanical Engineers, Part I: J. Systems and Control Engineering*, vol. 214, no. 2, pp. 87–97, 2000.

Application Library path: `MEMS_Module/Piezoelectric_Devices/piezoceramic_tube`

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 **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Structural Mechanics>Piezoelectric Devices**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

6 Click **Done**.

GEOMETRY 1

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**.

Create the tube by adding an off-axis rectangle in the axisymmetric geometry.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `240`.

4 In the **Height** text field, type `620`.

5 Locate the **Position** section. In the **r** text field, type `380`.

6 Click **Build All Objects**.

Add a PZT 5H to the model.

ADD MATERIAL

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

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

3 In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-5H)**.

4 Click **Add to Component** in the window toolbar.

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

SOLID MECHANICS (SOLID)

Piezoelectric Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Piezoelectric Material 1**.

2 In the **Settings** window for **Piezoelectric Material**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Material ZX-Plane System (comp1_zx_sys)**.

By selecting the material orientation as the zx-plane, you rotate the material so that its z direction is aligned with the r direction of the model, and the material's x direction is aligned with the model's z direction.

This example comprises two studies: the direct effect and inverse effect. All loadings for both studies are defined together and then a selection of relevant features will be done in the study settings.

Add a pressure follower load to the inner surface of the cylinder.

Boundary Load 1

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Boundary Load**.

2 Select Boundary 1 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 `0.1[MPa]`.

Constrain the lower surface of the tube with a roller boundary condition.

Roller 1

1 Right-click **Solid Mechanics (solid)** and choose **Roller**.

2 Select Boundary 2 only.

ELECTROSTATICS (ES)

Ground both the inner and outer surfaces of the cylinder.

Ground 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electrostatics (es)** and choose **Ground**.

2 Select Boundaries 1 and 4 only.

Add an electric potential feature on the outer boundary. This will over-ride the existing **Ground** feature.

Electric Potential 1

1 In the **Model Builder** window, right-click **Electrostatics (es)** and choose **Electric Potential**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

4 In the V_0 text field, type `1`.

MESH 1

Create a mapped mesh.

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, click **Build All**.

STUDY 1

The first study simulates the direct effect. All mechanical loads are kept and the electric potential feature is disabled in solver settings. It is automatically replaced by the ground feature that was previously overridden.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3 Select the **Modify physics tree and variables for study step** check box.
4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Electrostatics (es)>Electric Potential 1**.
5 Click **Disable**.
6 On the **Home** toolbar, click **Compute**.

RESULTS

Stress (solid)

The default plot groups show stress in the tube and the induced electric potential. Adapt these for comparison with . First replace stress plot by radial displacement.

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
2 In the **Settings** window for **2D Plot Group**, type `Radial Displacement (Direct Effect)` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Radial Displacement (Direct Effect)** node, then click **Surface 1**.
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>Solid Mechanics>Displacement>Displacement field (material and geometry frames)>u - Displacement field, R component**.
3 On the **Radial Displacement (Direct Effect)** toolbar, click **Plot**.

Stress, 3D (solid)

1 In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.
2 In the **Settings** window for **3D Plot Group**, type `Stress, 3D (Direct Effect)` in the **Label** text field.

Electric Potential (es)

1 In the **Model Builder** window, under **Results** click **Electric Potential (es)**.

2 In the **Settings** window for **2D Plot Group**, type `Electric Potential (Direct Effect)` in the **Label** text field.

Electric Potential, Revolved Geometry (es)

Change the data set of the potential plot in order to see a 3D cut view of the potential.

1 In the **Model Builder** window, under **Results** click **Electric Potential, Revolved Geometry (es)**.

2 In the **Settings** window for **3D Plot Group**, type `Electric Potential, 3D (Direct Effect)` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Revolution 2D 1**.

4 On the **Electric Potential, 3D (Direct Effect)** toolbar, click **Plot**.

Data Sets

Create a cross section through the geometry to use for line plots of the electric potential and displacement.

Cut Line 2D 1

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, set **r** to `380` and **z** to `300`.

4 In row **Point 2**, set **r** to `620` and **z** to `300`.

Visualize the cross section line.

5 Click **Plot**.

Add line plots of the radial displacement and the potential along the cross section.

1D Plot Group 5

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

2 In the **Settings** window for **1D Plot Group**, type `Radial Displacement, cut (Direct Effect)` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 1**.

Line Graph 1

1 Right-click **Radial Displacement, cut (Direct Effect)** and choose **Line Graph**.

2 In the **Model Builder** window, click **Line Graph 1**.

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>Solid Mechanics> Displacement>Displacement field (material and geometry frames)>u - Displacement field, R component**.

4 Locate the **y-Axis Data** section. From the **Unit** list, choose **nm**.

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

6 In the **Expression** text field, type `r`.

7 On the **Radial Displacement, cut (Direct Effect)** toolbar, click **Plot**.

Radial Displacement, cut (Direct Effect) 1

1 In the **Model Builder** window, under **Results** right-click **Radial Displacement, cut (Direct Effect)** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Voltage, Cut (Direct Effect)` in the **Label** text field.

Line Graph 1

1 In the **Model Builder** window, expand the **Results>Voltage, Cut (Direct Effect)** node, then click **Line Graph 1**.

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

3 In the **Expression** text field, type `abs(V)`.

4 On the **Voltage, Cut (Direct Effect)** toolbar, click **Plot**.

Finally add a new study to compute the results for the inverse effect.

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

The second study simulates the inverse effect. All electrical loads are kept and the pressure load feature is disabled in solver settings. It is automatically replaced by the **Free** boundary feature that was previously overridden.

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 **Physics and Variables Selection** section.

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Boundary Load 1**.

5 Click **Disable**.

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

RESULTS

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Radial Displacement (Inverse Effect)` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Radial Displacement (Inverse Effect)** node, then click **Surface 1**.

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>Solid Mechanics>Displacement>Displacement field (material and geometry frames)>u - Displacement field, R component**.

3 On the **Radial Displacement (Inverse Effect)** toolbar, click **Plot**.

Stress, 3D (solid)

1 In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.

2 In the **Settings** window for **3D Plot Group**, type `Stress, 3D (Inverse Effect)` in the **Label** text field.

Electric Potential (es)

1 In the **Model Builder** window, under **Results** click **Electric Potential (es)**.

2 In the **Settings** window for **2D Plot Group**, type `Electric Potential (Inverse Effect)` in the **Label** text field.

Electric Potential, Revolved Geometry (es)

1 In the **Model Builder** window, under **Results** click **Electric Potential, Revolved Geometry (es)**.

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

3 From the **Data set** list, choose **Revolution 2D 3**.

4 In the **Label** text field, type `Electric Potential, 3D (Inverse Effect)`.

5 On the **Electric Potential, 3D (Inverse Effect)** toolbar, click **Plot**.

Create a second **Cut Line 2D** for the new solution.

Cut Line 2D 2

1 In the **Model Builder** window, under **Results>Data Sets** right-click **Cut Line 2D 1** and choose **Duplicate**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Data** section.

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

Radial Displacement, cut (Direct Effect) 1

1 In the **Model Builder** window, under **Results** right-click **Radial Displacement, cut (Direct Effect)** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Radial Displacement, cut (Inverse Effect)` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 2**.

4 On the **Radial Displacement, cut (Inverse Effect)** toolbar, click **Plot**.

Voltage, Cut (Direct Effect) 1

1 In the **Model Builder** window, under **Results** right-click **Voltage, Cut (Direct Effect)** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Voltage, Cut (Inverse Effect)` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 2**.

4 On the **Voltage, Cut (Inverse Effect)** toolbar, click **Plot**.

Piezoelectric Valve

Introduction

Piezoelectric valves are frequently employed in medical and laboratory applications. They have a number of advantages over competing technologies, including minimal heat dissipation, quiet operation, energy efficiency, durability, low weight and fast response times. These valves typically consist of a seal that is pushed up against an opening to close the valve, or moved away from the opening to open the valve, by a piezoelectric actuator. The actuator itself often has a complex internal structure, with stacked layers of piezoelectric separated by thin conducting layers that are connected together in such a way that the applied field leads to a large deformation.

This model shows how to model a piezoelectric valve in COMSOL. The valve is actuated by a stacked piezoelectric bimorph disc actuator, which compresses a hyper-elastic seal against the valve opening to shut off the flow. The detailed construction and operation of the stacked actuator is considered in the model.

Note: This application requires the Nonlinear Structural Materials Module.

Model Definition

Figure 1 shows both an axisymmetric slice through the geometry and a 3D rendering of the geometry. In this simple valve design a disc actuator compresses a hyperelastic seal directly onto an annular opening in a stainless steal support structure. The construction of the actuator itself is illustrated in Figure 2. The outer edge of the disc annulus is clamped to a stainless steel base and supporting structure. When a voltage is applied to the actuator the disc bends causing a vertical motion of the central opening of the annular actuator. With an appropriate polarity the opening moves downward, towards an annular opening in the base (supported at regular intervals by struts not included in the model). As the actuator moves towards the opening a hyperelastic seal is compressed against a mating structure, sealing up the opening. Within the model, the contact between the seal and the mating structure is modeled in detail, as is the operation of the actuator.

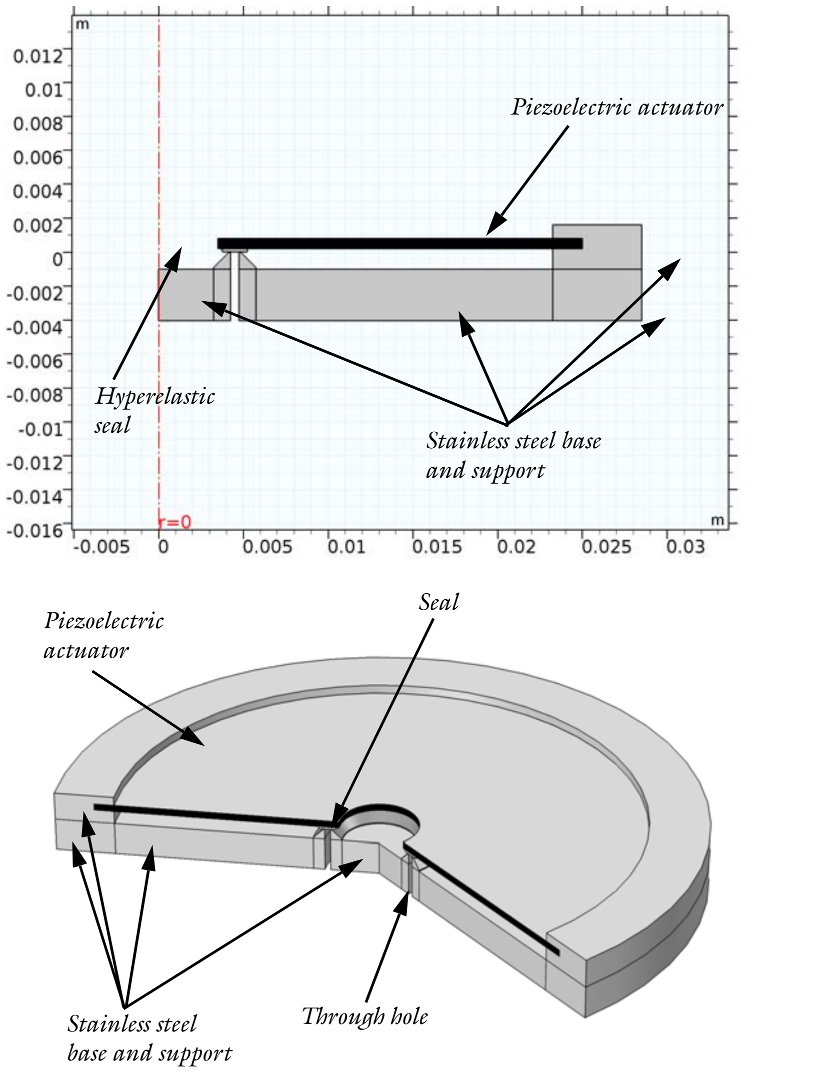


Figure 1: Axisymmetric model geometry (top) and full 3D geometry (bottom). Key components of the geometry are labeled.

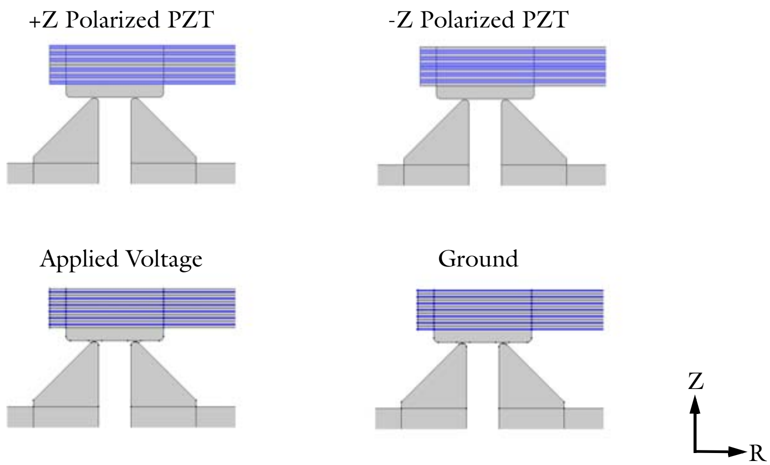


Figure 2: Detail of the actuator and seal region, showing the construction of the actuator itself. The actuator consists of layers of PZT with metal layers between the various layers. Alternate metal layers are connected to ground whilst a voltage is applied to the other layers. Alternate layers of PZT are polarized in opposite directions. Two such actuators are stacked in such a manner that the applied potential causes contraction of one half of the beam and expansion of the other half. This results in a net bending moment acting on the beam.

Results and Discussion

Figure 3 shows the strain in the hyperelastic seal when the applied voltage is 60 V. The strain is localized in the vicinity of the contact region. Figure 4 shows the von Mises strain in the piezoelectric and its supporting structures at the same applied voltage. The stress is maximal in the PZT close to the contact. The potential within the actuator is shown in Figure 5. It is clear that the applied potentials match those shown in Figure 2. Finally the contact pressure is shown in Figure 6. The maximum pressure is 6×10^5 Pa on the surface of the seal that separates the inlet of the valve from the outlet.

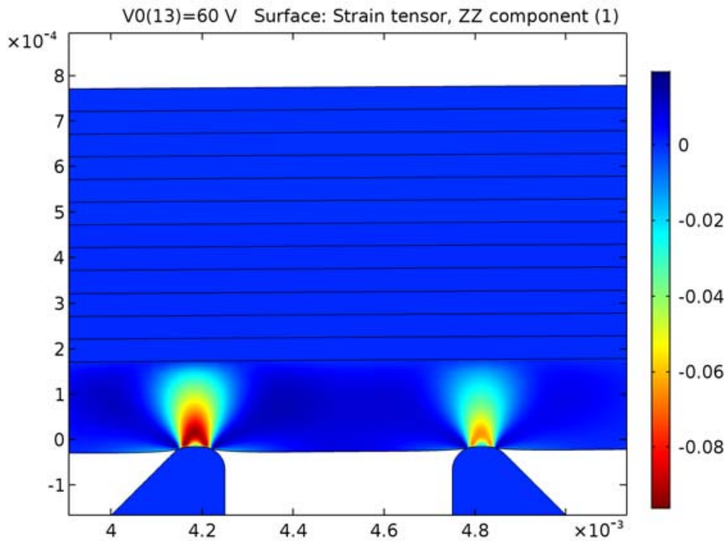


Figure 3: ZZ component of the strain in the vicinity of the contact.

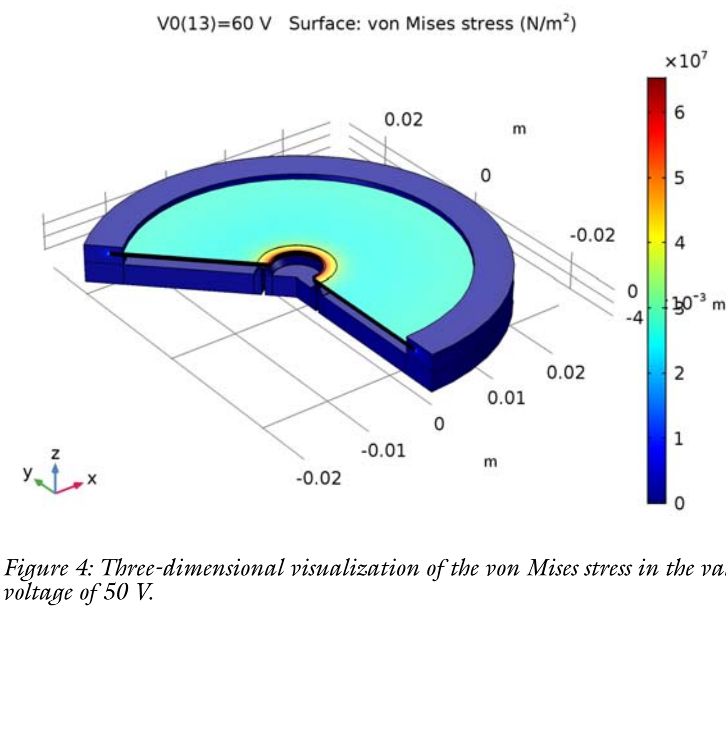


Figure 4: Three-dimensional visualization of the von Mises stress in the valve at an applied voltage of 50 V.

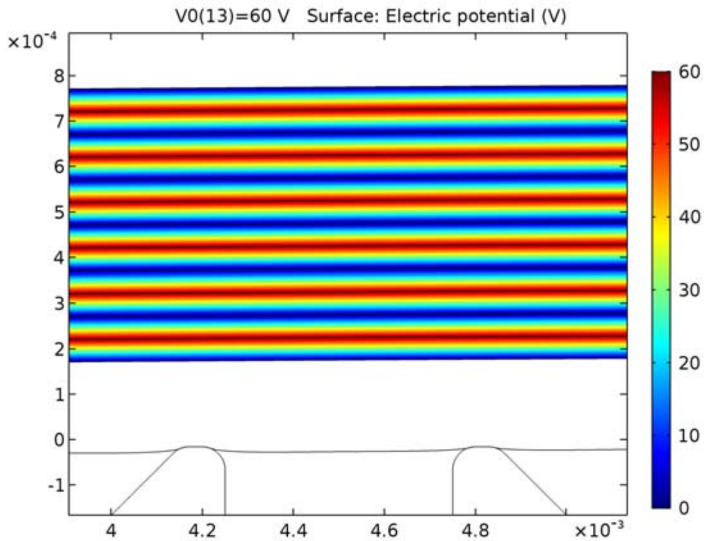


Figure 5: Electric potential inside the actuator in the vicinity of the contact.

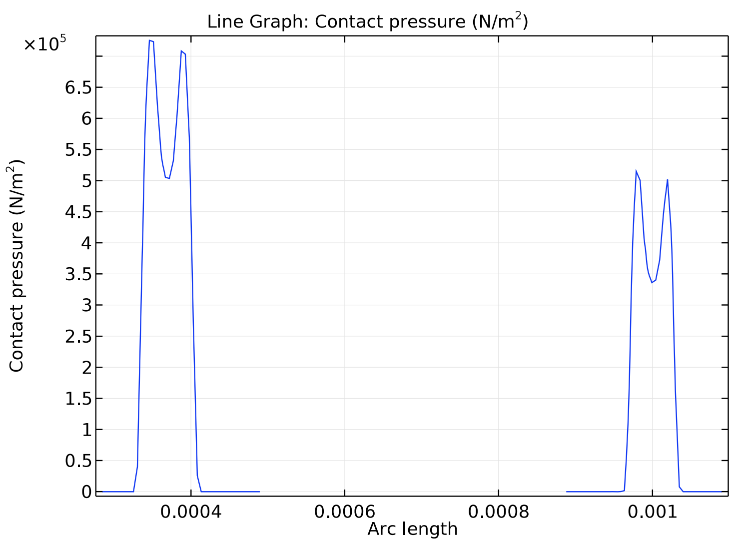


Figure 6: Contact pressure as a function of position along the surface of the seal.

Application Library path: `MEMS_Module/Piezoelectric_Devices/piezoelectric_valve`

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 **2D Axisymmetric**.
2. In the **Select Physics** tree, select **Structural Mechanics>Piezoelectric Devices**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
6. Click **Done**.

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

GEOMETRY 1

1. On the **Geometry** toolbar, click **Insert Sequence**.
2. Browse to the model's Application Libraries folder and double-click the file `piezoelectric_valve_geom_sequence.mph`.
3. On the **Geometry** toolbar, click **Build All**.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, expand the **Global Definitions** node, then click **Parameters**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
`V0`	`50[V]`	50 V	`Applied voltage`

DEFINITIONS

Explicit 1

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

2 Select Domains 5, 7, 9, 10, 12, 14, 18, 20, 22, 23, 25, 27, 32, 34, 36, 37, 39, 41, 47, 49, 51, 52, 54, and 56 only.

3 In the **Settings** window for **Explicit**, type `+ Polarized` in the **Label** text field.

Explicit 2

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

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

3 Select Domains 4, 6, 8, 11, 13, 15, 17, 19, 21, 24, 26, 28, 31, 33, 35, 38, 40, 42, 46, 48, 50, 53, 55, and 57 only.

Union 1

1 On the **Definitions** toolbar, click **Union**.

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

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, In the **Selections to add** list, choose **+ Polarized** and **- Polarized**.

5 Click **OK**.

Explicit 3

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

2 On the **Definitions** toolbar, click **Explicit**.

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

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

5 In the **Label** text field, type `Ground`.

6 Select Boundaries 10, 14, 18, 22, 26, 30, 33, 36, 40, 44, 48, 52, 56, 59, 76, 80, 84, 88, 92, 96, 99, 109, 113, 117, 121, 125, 129, and 133 only.

Explicit 4

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

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

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

4 Select Boundaries 12, 16, 20, 24, 28, 32, 38, 42, 46, 50, 54, 58, 78, 82, 86, 90, 94, 98, 111, 115, 119, 123, 127, and 131 only.

Explicit 5

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Explicit 5**.

2 In the **Settings** window for **Explicit**, type `Mapped Mesh Steel` in the **Label** text field.

3 Select Domains 1, 2, 29, 43, and 44 only.

Union 2

1 On the **Definitions** toolbar, click **Union**.

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

3 Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, In the **Selections to add** list, choose **Piezoelectric** and **Mapped Mesh Steel**.

5 Click **OK**.

6 In the **Settings** window for **Union**, type `Mapped Mesh` in the **Label** text field.

Contact Pair 1 (p1)

1 On the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 Select Boundaries 8, 62, 71, 72, and 150–153 only.

3 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

4 Select the **Active** toggle button.

5 Select Boundaries 61 and 66 only.

Material XZ-Plane System (comp1_xz_sys)

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Material XZ-Plane System (comp1_xz_sys)**.

2 In the **Settings** window for **Base Vector System**, type `+Z Polarized` in the **Label** text field.

+Z Polarized 1 (comp1_xz_sys1)

1 Right-click **Component 1 (comp1)>Definitions>+Z Polarized** and choose **Duplicate**.

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

3 Find the **Base vectors** subsection. In the table, enter the following settings:

	r	z
x1	1	0
x3	0	-1

4 In the **Label** text field, type `-Z Polarized`.

SOLID MECHANICS (SOLID)

In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)** node.

Piezoelectric Material 2

1 Right-click **Solid Mechanics (solid)** and choose **Material Models>Piezoelectric Material**.

2 In the **Settings** window for **Piezoelectric Material**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **-Z Polarized (comp1_xz_sys1)**.

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

Piezoelectric Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Piezoelectric Material 1**.

2 In the **Settings** window for **Piezoelectric Material**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **+ Polarized**.

Hyperelastic Material 1

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Material Models>Hyperelastic Material**.

2 Select Domain 16 only.

3 In the **Settings** window for **Hyperelastic Material**, locate the **Hyperelastic Material** section.

4 From the **Material model** list, choose **Mooney-Rivlin, two parameters**.

5 In the κ text field, type `1e4[MPa]`.

Fixed Constraint 1

1 Right-click **Solid Mechanics (solid)** and choose **Domain Constraints>Fixed Constraint**.

2 Select Domains 1, 2, 29, 43, and 44 only.

Contact 1

1 Right-click **Solid Mechanics (solid)** and choose **Pairs>Contact**.

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

3 In the **Pairs** list, select **Contact Pair 1 (p1)**.

ELECTROSTATICS (ES)

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

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

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

Terminal 1

1 Right-click **Component 1 (comp1)>Electrostatics (es)** and choose the boundary condition **Terminal**.

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

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

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

5 In the V_0 text field, type `V0`.

Ground 1

1 In the **Model Builder** window, right-click **Electrostatics (es)** and choose **Ground**.

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

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

ADD MATERIAL

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

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

3 In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-5H)**.

4 Click **Add to Component 1**.

MATERIALS

Lead Zirconate Titanate (PZT-5H) (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Lead Zirconate Titanate (PZT-5H) (mat1)**.

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

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

ADD MATERIAL

1 Go to the **Add Material** window.

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

3 Click **Add to Component 1**.

MATERIALS

Steel AISI 4340 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Steel AISI 4340 (mat2)**.

2 Select Domains 1–3, 29, 30, and 43–45 only.

Material 3 (mat3)

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

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

3 Select Domain 16 only.

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

Property	Name	Value	Unit	Property group
Density	rho	`1800[kg/m^3]`	kg/m³	Basic
Model parameters	C10	`0.37[MPa]`	Pa	Mooney-Rivlin
Model parameters	C01	`0.11[MPa]`	Pa	Mooney-Rivlin

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

MESH 1

Edge 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Edge**.

2 Select Boundaries 61, 62, 66, 71, and 150–153 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Edge 1** and choose **Size**.

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

3 Click **Clear Selection**.

4 Select Boundaries 61 and 66 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

6 Click the **Custom** button.

7 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

8 In the associated text field, type `w0/100`.

Size 2

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

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

3 Click **Clear Selection**.

4 Select Boundaries 62, 71, and 150–153 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

6 Click the **Custom** button.

7 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

8 In the associated text field, type `w0/50`.

Size

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

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

3 From the **Predefined** list, choose **Extremely fine**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `w0`.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

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

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

4 From the **Selection** list, choose **Mapped Mesh**.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Boundaries 75, 77, 79, 81, 83, 85, 87, 89, 91, 93, 95, and 97 only.

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

4 In the **Number of elements** text field, type `2`.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 36 and 109 only.

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

4 In the **Number of elements** text field, type 40.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

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

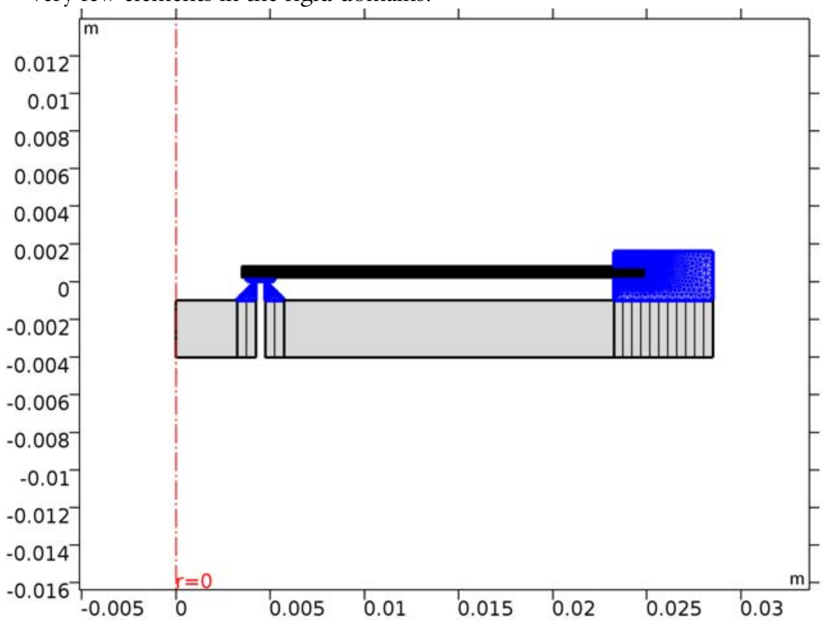
3 In the **Number of elements** text field, type 1.

4 Select Boundaries 2, 63, 67, and 101 only.

5 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Triangular**.

6 In the **Settings** window for **Mesh**, click **Build All**.

Compare the resulting mesh with that shown below. Note that there are purposefully very few elements in the rigid domains.



STUDY 1

Step 1: Stationary

1 In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.

2 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V0		

5 Click **Range**.

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

7 In the **Step** text field, type `5`.

8 In the **Stop** text field, type `60`.

9 Click **Add**.

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

11 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
V0	range(0,5,60)	V

When modeling contact both the contact pressure and the auxiliary pressure need to be manually scaled. It is good practice to modify the manual scaling of these variables to an appropriate value.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

The contact pressure is expected to be of the order of MPa, so set the scales accordingly.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Auxiliary pressure (comp1.solid.pw)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 In the **Scale** text field, type `1e6`.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** click **Contact pressure (comp1.solid.Tn_p1)**.

7 In the **Settings** window for **Field**, locate the **Scaling** section.

8 In the **Scale** text field, type `1e6`.

9 On the **Study** toolbar, click **Compute**.

RESULTS

2D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
2 In the **Settings** window for **2D Plot Group**, type `Strain (ZZ component)` in the **Label** text field.
3 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (r, phi, z)**.

Surface 1

1 Right-click **Strain (ZZ component)** 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>Solid Mechanics>Strain> Strain tensor (material and geometry frames)>solid.eZZ - Strain tensor, ZZ component**.
3 Locate the **Coloring and Style** section. Select the **Reverse color table** check box.
4 On the **Strain (ZZ component)** toolbar, click **Plot**.

Zoom in on the contact region and compare the plot with .

5 Click the **Zoom Box** button on the **Graphics** toolbar.

Strain (ZZ component) 1

1 In the **Model Builder** window, under **Results** right-click **Strain (ZZ component)** and choose **Duplicate**.
2 In the **Settings** window for **2D Plot Group**, type `Electric Field (Z component)` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Strain (ZZ component) 1** node, then click **Results>Electric Field (Z component)>Surface 1**.
2 In the **Settings** window for **Surface**, click to collapse the **Expression** section.
3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electrostatics>Electric> Electric field (material and geometry frames)>es.EZ - Electric field, Z component**.
4 On the **Electric Field (Z component)** toolbar, click **Plot**.

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**, type `Contact Pressure` in the **Label** text field.
3 Locate the **Data** section. From the **Parameter selection (V0)** list, choose **Last**.

Line Graph 1

1. Right-click **Contact Pressure** and choose **Line Graph**.
2. Select Boundaries 60, 61, 65, 66, and 73 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>Solid Mechanics> Contact>solid.Tn - Contact pressure**.
4. On the **Contact Pressure** toolbar, click **Plot**.

Appendix — Geometry Instructions

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

NEW

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

MODEL WIZARD

1. In the **Model Wizard** window, click **2D Axisymmetric**.
2. Click **Done**.

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:

Name	Expression	Value	Description
t0	0.05[mm]	5E-5 m	Piezoelectric layer thickness
ID	7[mm]	0.007 m	Disc actuator inner diameter
OD	50[mm]	0.05 m	Disc actuator outer diameter
n	12	12	Number of layers in actuator
ts	0.2[mm]	2E-4 m	Thickness of seal
w0	0.5[mm]	5E-4 m	Through hole dimension
w1	ID/2	0.0035 m	Clamp region dimension
w2	ID/4	0.00175 m	Overall clamp dimension

Name	Expression	Value	Description
h0	5*t0*n	0.003 m	Base thickness
deltaz	16[um]	1.6E-5 m	Contact offset at 0[V]

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `(OD-ID)/2`.

4 In the **Height** text field, type `t0`.

5 Locate the **Position** section. In the **r** text field, type `ID/2`.

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

7 Click to expand the **Layers** section. Select the **Layers to the left** check box.

8 Clear the **Layers on bottom** check box.

9 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.5*w0
Layer 2	3*w0

Array 1 (arr1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

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

3 In the **z size** text field, type `n`.

4 Locate the **Displacement** section. In the **z** text field, type `t0`.

5 Select the object **r1** only.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `3*w0`.

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

5 Locate the **Position** section. In the **r** text field, type `ID/2+0.5*w0`.

Fillet 1 (fil1)

1 On the **Geometry** toolbar, click **Fillet**.

2 On the object **r2**, select Points 1 and 2 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `ts/3`.

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `2*w0`.

4 In the **Height** text field, type `2*w0+h0`.

5 Locate the **Position** section. In the **r** text field, type `ID/2-0.5*w0`.

6 In the **z** text field, type `-2*w0-deltaz-h0`.

Chamfer 1 (cha1)

1 On the **Geometry** toolbar, click **Chamfer**.

2 On the object **r3**, select Point 4 only.

3 In the **Settings** window for **Chamfer**, locate the **Distance** section.

4 In the **Distance from vertex** text field, type `1.8*w0`.

Fillet 2 (fil2)

1 On the **Geometry** toolbar, click **Fillet**.

2 On the object **cha1**, select Points 3 and 5 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `0.1*w0`.

Polygon 1 (pol1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the **r** text field, type `ID/2+1.2*w0 ID/2+1.6*w0`.

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

Mirror 1 (mir1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Select the objects **pol1** and **fil2** only.

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

4 Select the **Keep input objects** check box.

5 Locate the **Point on Line of Reflection** section. In the **r** text field, type `ID/2+2*w0`.

Rectangle 4 (r4)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `ID/2+1.5*w0`.

4 In the **Height** text field, type `h0`.

5 Locate the **Position** section. In the **z** text field, type `-2*w0-deltaz-h0`.

Rectangle 5 (r5)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `OD/2+w1-ID/2-2.5*w0`.

4 In the **Height** text field, type `h0`.

5 Locate the **Position** section. In the **r** text field, type `ID/2+2.5*w0`.

6 In the **z** text field, type `-2*w0-deltaz-h0`.

Rectangle 6 (r6)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `w2+w1`.

4 In the **Height** text field, type `h0+4*w0+n*t0+deltaz`.

5 Locate the **Position** section. In the **r** text field, type `OD/2-w2`.

6 In the **z** text field, type `-2*w0-deltaz-h0`.

7 Click **Build Selected**.

Created in COMSOL Multiphysics 5.3

Piezoresistive Pressure Sensor

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).

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 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 geometry, since 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 model. The piezoresistor is oriented at 45 to the die edge, and so lies in the [100] direction of the crystal. 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.

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

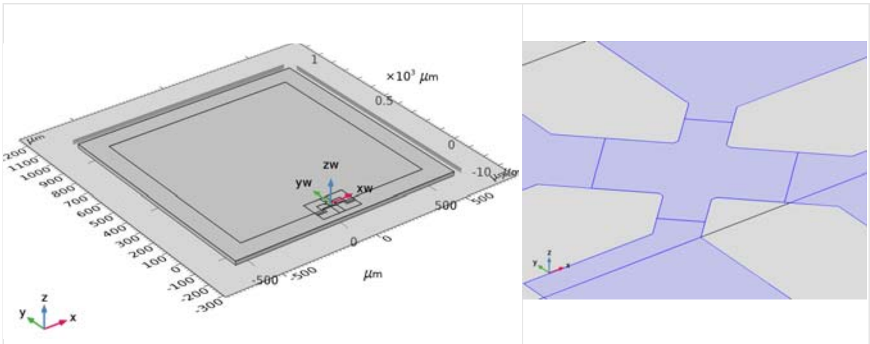


Figure 1: Left: Model geometry. Right: Detail showing the piezoresistor geometry.

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, $\boldsymbol{E}$, 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 COMSOL's 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 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 employed by COMSOL 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 $\Delta\rho$ vector computed from Equation 3 is assembled into matrix form in the following manner in Equation 1:

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

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] orientated 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.

COMSOL’s Piezoresistivity interfaces solve Equation 3 and an inverse form of Equation 4, together with the equations of structural mechanics. In this model the *Piezoresistivity, 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. At the center of the diaphragm the displacement is 1.2 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 3, which is also 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 at the center of each of the two edges along which the plot is made, but the value of this maximum is less than the maximum stress in the model, in part due to the boundary conditions employed on the three dimensional diaphragm. The model:

piezoresistive_pressure_sensor_shell.mph shows better agreement with the theoretical maximum shear stress along this edge.

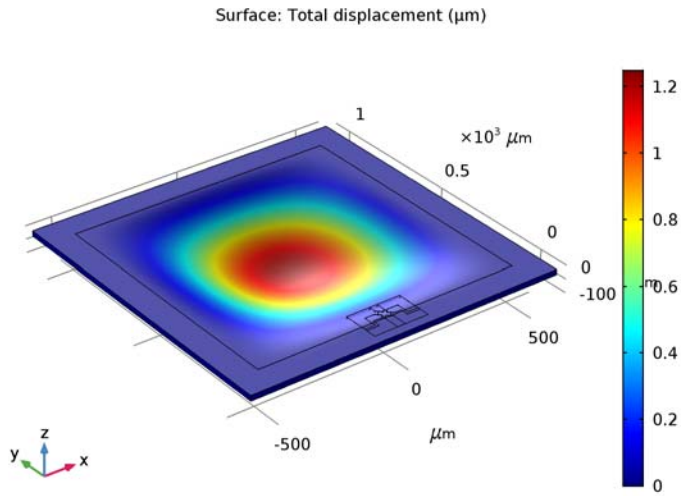


Figure 2: Diaphragm displacement as a result of a 100 kPa applied pressure.

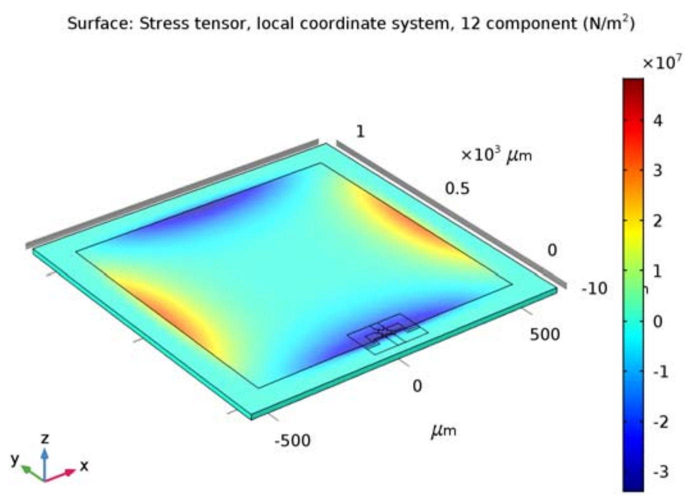


Figure 3: Shear stress, shown in the local co-ordinate system of the piezoresistor (rotated 45° about the z-axis of the global system). The shear stress is has its highest magnitude close to the piezoresistor with a value of approximately -35 MPa.

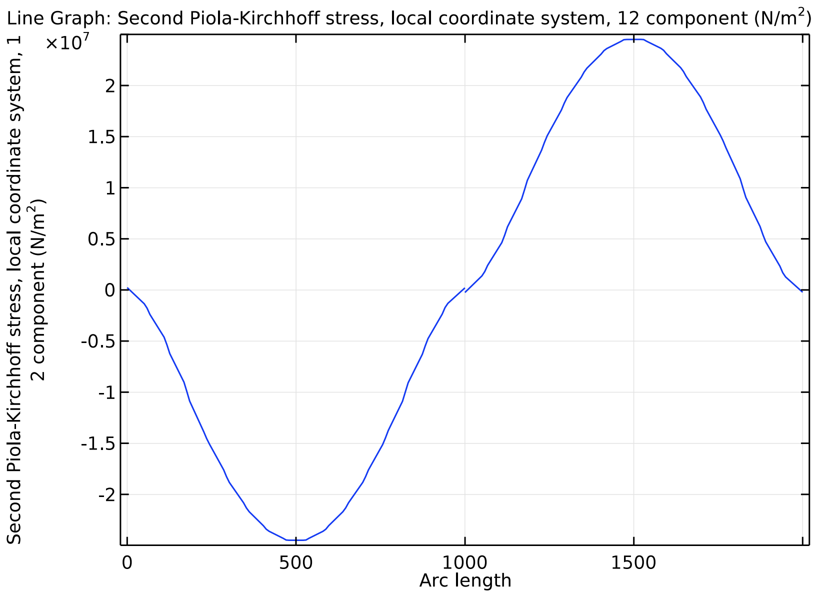


Figure 4: Plot of the local shear stress along two edges of the diaphragm.

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 (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 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.

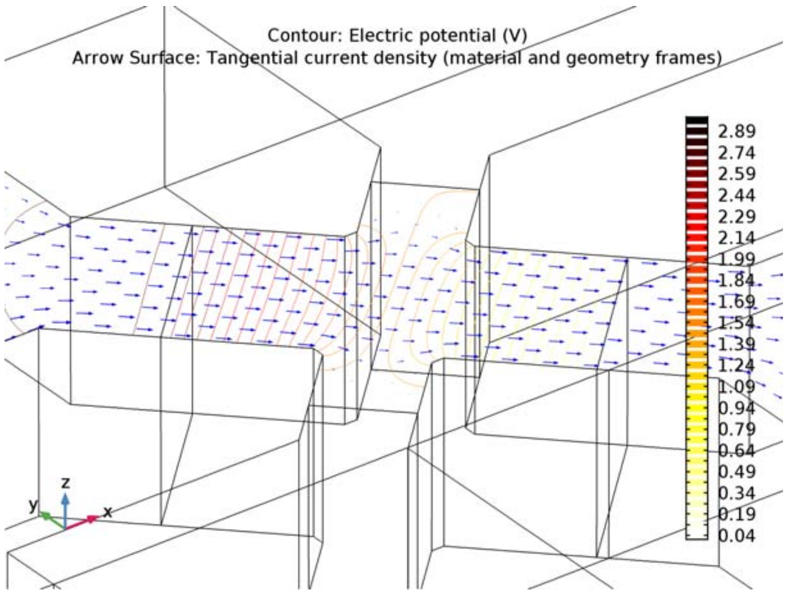


Figure 5: Arrows: Current density, Contours: Electric Potential, for a device driven by a 3 V bias with an applied pressure of 100 kPa.

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`

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>Piezoresistivity>Piezoresistivity, Boundary Currents**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
6. Click **Done**.

GEOMETRY 1

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 .
4. On the **Geometry** toolbar, click **Insert Sequence**.
5. Browse to the model's Application Libraries folder and double-click the file `piezoresistive_pressure_sensor_geom_sequence.mph`.
6. On the **Geometry** toolbar, click **Build All**.

DEFINITIONS

Explicit 1

1. On 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 46 only.

Explicit 2

1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Connections` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 14, 22, 26, 39, 46, 73, 77, 81, and 104 only.

Box 1

1 On the **Definitions** toolbar, click **Box**.
2 In the **Settings** window for **Box**, type `Membrane (Lower Surface)` 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 In the **z maximum** text field, type `-1`.
9 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Membrane (Lower Surface) 1

1 Right-click **Membrane (Lower Surface)** and choose **Duplicate**.
2 In the **Settings** window for **Box**, type `Membrane (Upper Surface)` in the **Label** text field.
3 Locate the **Box Limits** section. In the **z minimum** text field, type `-1`.
4 In the **z maximum** text field, type `inf`.

Box 3

1 On the **Definitions** toolbar, click **Box**.
2 In the **Settings** window for **Box**, type `Lower Surface` 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 **z maximum** text field, type `-1`.
5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Lower Surface 1

1 Right-click **Lower Surface** and choose **Duplicate**.
2 In the **Settings** window for **Box**, type `Upper Surface` in the **Label** text field.
3 Locate the **Box Limits** section. In the **z minimum** text field, type `-1`.
4 In the **z maximum** text field, type `inf`.

Difference 1

1 On the **Definitions** toolbar, click **Difference**.

2 In the **Settings** window for **Difference**, type `Fixed` 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, select **Lower Surface** 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 (Lower Surface)** in the **Selections to subtract** list.

10 Click **OK**.

Union 1

1 On the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, 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. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, In the **Selections to add** list, choose **Piezoresistor** and **Connections**.

6 Click **OK**.

Rotated System 2 (sys2)

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

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

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

ADD MATERIAL

1 On 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 1**.

ADD MATERIAL

1 Go to the **Add Material** window.

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

3 Click **Add to Component 1**.

MATERIALS

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

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

2 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **p-Silicon (single-crystal, lightly doped) (mat2)**.

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

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

5 From the **Selection** list, choose **Electric Currents**.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

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

3 From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

4 Locate the **Linear Elastic Material** section. From the **Solid model** list, choose **Anisotropic**.

5 From the **Material data ordering** list, choose **Voigt (XX, YY, ZZ, YZ, XZ, XY)**.

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

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

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

Boundary Load 1

1 Right-click **Solid Mechanics (solid)** and choose **Boundary Load**.

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

3 From the **Selection** list, choose **Membrane (Upper Surface)**.

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

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

ELECTRIC CURRENTS, SHELL (ECS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents, Shell (ecs)**.

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

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

4 Locate the **Shell Thickness** section. In the d_s text field, type `400[nm]`.

Current Conservation 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electric Currents, Shell (ecs)** click **Current Conservation 1**.

2 In the **Settings** window for **Current Conservation**, locate 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 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electric Currents, Shell (ecs)** 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 `1.32e19[1/cm^3]`.

5 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

Ground 1

1 In the **Model Builder** window, right-click **Electric Currents, Shell (ecs)** and choose **Ground**.

2 Select Edge 195 only.

Terminal 1

1 Right-click **Electric Currents, Shell (ecs)** and choose **Terminal**.

2 Select Edges 30 and 35 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 Right-click **Electric Currents, Shell (ecs)** and choose **Terminal**.

2 Select Edge 20 only.

Terminal 3

1 Right-click **Electric Currents, Shell (ecs)** and choose **Terminal**.

2 Select Edges 201 and 205 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 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.

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** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `2`.

8 Select the **Minimum element size** check box.

9 In the associated text field, type `0.1`.

Size 2

1 Right-click **Component 1 (comp1)>Mesh 1>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 **Component 1 (comp1)>Mesh 1>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 Click the **Select Box** button on the **Graphics** toolbar.

5 Select Edges 74, 79, 104, 108, 111, 114, 117, 120, 123, 142, 143, and 146 only.

6 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.4`.

Free Tetrahedral 1

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

Free Triangular 1

1 Right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Upper Surface**.

Swept 1

1 Right-click **Mesh 1** and choose **Swept**.

2 On the **Home** toolbar, click **Build Mesh**.

3 Click **Compute**.

RESULTS

Stress (solid)

The default plots show the von Mises stress and the electric potential. Now create a plot of the total displacement to compare with .

3D Plot Group 3

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

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

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>Solid Mechanics>Displacement>solid.disp - Total displacement**.

Deformation 1

1 Right-click **Results>Displacement>Surface 1** and choose **Deformation**.

2 On the **Displacement** toolbar, click **Plot**.

Now create a plot of the shear stress in the local coordinate system of the piezoresistor, to compare with .

3D Plot Group 4

1 On 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.

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>Solid Mechanics>Stress>Stress tensor, local coordinate system>solid.sl12 - Stress tensor, local coordinate system, 12 component**.

3 On the **In-Plane Shear Stress (Local Coordinates)** toolbar, click **Plot**.

Create a line plot of the shear stress in the local coordinate system of the piezoresistor, to compare with .

1D Plot Group 5

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

2 In the **Settings** window for **1D 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 16 and 213 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>Solid Mechanics>Stress>Second Piola-Kirchhoff stress, local coordinate system>solid.Sl12 - Second Piola-Kirchhoff stress, local coordinate system, 12 component**.

4 On 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 .

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Data Sets** node.

Selection

1 Right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

2 On the **Results** toolbar, click **Selection**.

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

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

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

6 Click **Zoom to Selection**.

7 From the **Selection** list, choose **Electric Currents**.

3D Plot Group 6

1 On the **Results** toolbar, click **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 **Data set** list, choose **Study 1/Solution 1 (2) (sol1)**.

Contour 1

1 Right-click **Current and Voltage** and choose **Contour**.

2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electric Currents, Shell> Electric>V - Electric potential**.

3 Locate the **Levels** section. In the **Total levels** text field, type `40`.

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

5 Select the **Reverse color table** check box.

Arrow Surface 1

1 In the **Model Builder** window, under **Results** 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>Electric Currents, Shell>Currents and charge>ecs.tJX,...,ecs.tJZ - Tangential current density (material and geometry frames)**.

3 Locate the **Coloring and Style** section. Select the **Scale factor** check box.

4 In the associated text field, type `2e-9`.

5 In the **Number of arrows** text field, type `3000`.

6 From the **Color** list, choose **Blue**.

7 On the **Current and Voltage** toolbar, click **Plot**.

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

Global Evaluation 1

1 On 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> Electric Currents, Shell>Terminals>ecs.I0_1 - Terminal current**.

3 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Electric Currents, Shell>Terminals>ecs.V0_2 - Terminal voltage**.

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**.

Created in COMSOL Multiphysics 5.3

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

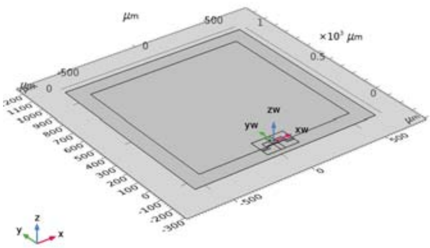
The model consists square membrane with side 1 mm and thickness 20 µm, supported around its edges by region 0.1mm wide, which is intended to represent the remainder of the wafer. The supporting region is fixed on its underside (representing a connection to the thicker handle of the device die). Near to one edge of the membrane an X-shaped piezoresistor (or Xducer^{TM})[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 1.32×10^{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 geometry, since 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 model. The piezoresistor is oriented at 45 to the die edge, and so lies in the [100] direction of the crystal. 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.

COMSOL model, a co-ordinate system rotated 45 about the global Z-axis is added to define the orientation of the crystal.



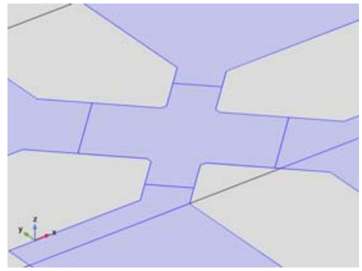


Figure 1: Left: Model geometry. Right: Detail showing the piezoresistor geometry.

DEVICE PHYSICS AND EQUATIONS

The conductivity of the Xducer$^{\text{TM}}$ 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:

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

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:

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

where Π is the piezoresistance tensor (SI units: $\text{Pa}^{-1}\Omega\text{m}$), a material property. Note that COMSOL's 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 employed by COMSOL 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 $\Delta\rho$ vector computed from Equation 3 is assembled into matrix form in the following manner in Equation 1:

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

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$^{\text{TM}}$ 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 Xducer$^{\text{TM}}$ 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.

COMSOL's Piezoresistance interfaces 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 mid point 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.

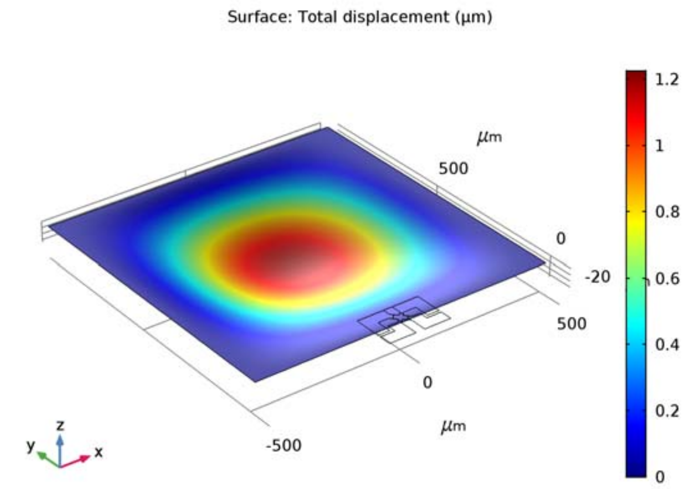


Figure 2: Diaphragm displacement as a result of a 100 kPa applied pressure.

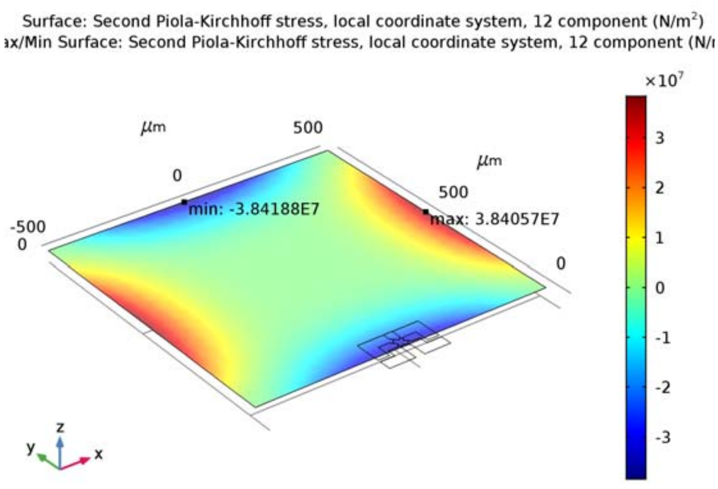


Figure 3: Shear stress, shown in the local co-ordinate system of the piezoresistor (rotated 45° about the z-axis of the global system). The shear stress is has its highest magnitude close to the piezoresistor with a value of approximately -35 MPa.

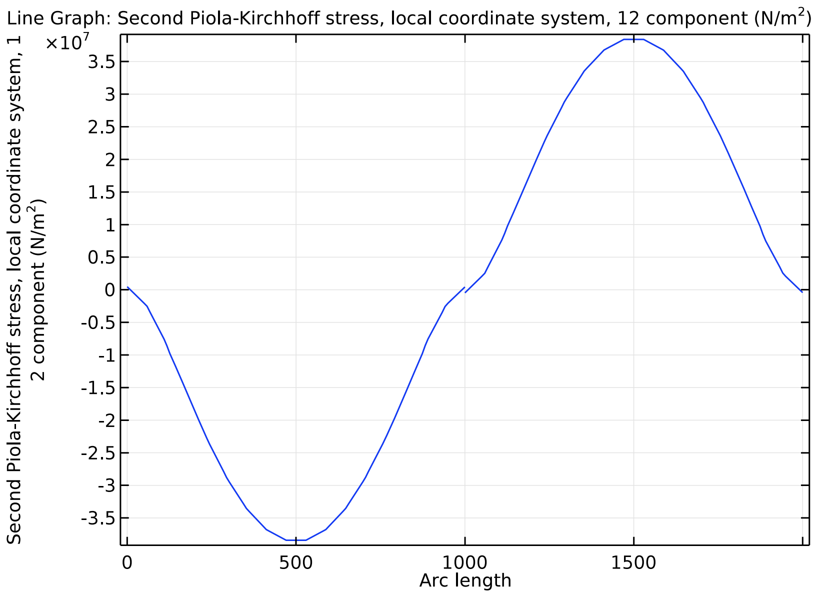


Figure 4: Plot of the local shear stress along two edges of the diaphragm.

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 Xducer$^{\mathrm{TM}}$ 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.

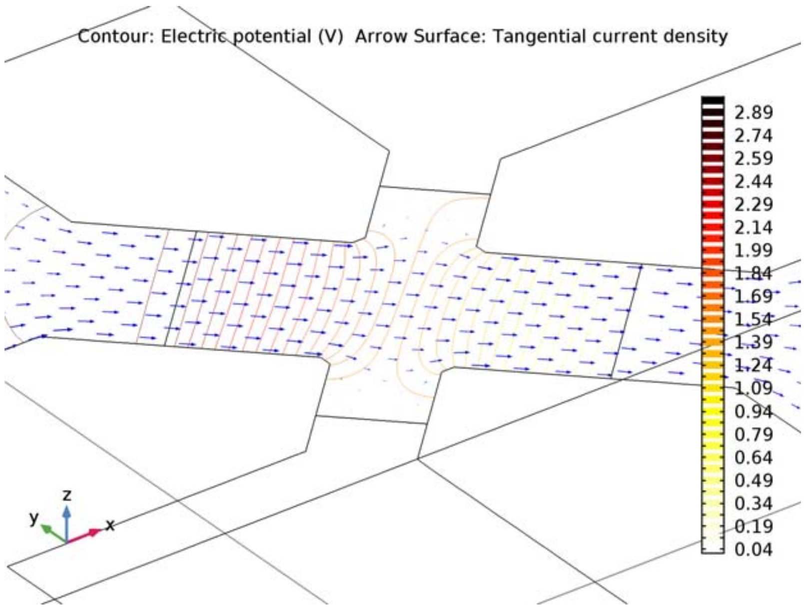


Figure 5: Arrows: Current density, Contours: Electric Potential, for a device driven by a 3 V bias with an applied pressure of 100 kPa.

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>Piezoresistivity>Piezoresistivity, Shell**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

6 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, 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 .

4 On 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 On 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

Explicit 1

1 On 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.

Piezoresistor 1

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.

Box 1

1 On 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**.

Union 1

1 On 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**.

Model boundaries 1

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**.

Adjacent 1

1 On 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**.

Difference 1

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

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

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

ADD MATERIAL

1 On 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.

ADD MATERIAL

1 Go to the **Add Material** window.

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

3 Click **Add to Component** in the window toolbar.

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

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

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **p-Silicon (single-crystal, lightly doped) (mat2)**.

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

3 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**.

4 Locate the **Thickness** section. In the d text field, type `20[um]`.

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** 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 (XX, YY, ZZ, YZ, XZ, XY)**.

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 **Model Builder** window, right-click **Shell (shell)** 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 Right-click **Shell (shell)** and choose **Face Constraints>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 1

1 Right-click **Shell (shell)** and choose **Face and Volume Loads>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, SHELL (ECS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents, Shell (ecs)**.

2 In the **Settings** window for **Electric Currents, 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]`.

Current Conservation 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electric Currents, Shell (ecs)** click **Current Conservation 1**.

2 In the **Settings** window for **Current Conservation**, locate 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 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electric Currents, Shell (ecs)** 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 `1.32e19[1/cm^3]`.

5 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

Ground 1

1 In the **Model Builder** window, right-click **Electric Currents, Shell (ecs)** and choose **Ground**.

2 Select Edge 70 only.

Terminal 1

1 Right-click **Electric Currents, Shell (ecs)** 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 Right-click **Electric Currents, Shell (ecs)** and choose **Terminal**.

2 Select Edge 7 only.

Terminal 3

1 Right-click **Electric Currents, Shell (ecs)** 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 **Component 1 (comp1)>Mesh 1>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 **Component 1 (comp1)>Mesh 1>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 **Component 1 (comp1)>Mesh 1>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, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 1** and choose **Move Down**.

2 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Move Down**.

3 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Move Down**.

4 On 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 total displacement to compare with .

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Data Sets** node.

Selection

1 Right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

2 On the **Results** toolbar, click **Selection**.

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

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

5 From the **Selection** list, choose **Membrane**.

3D Plot Group 5

1 On 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 **Data set** 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 **Model>Component 1>Shell> Displacement>shell.disp - Total displacement**.

Deformation 1

1 Right-click **Results>Displacement>Surface 1** and choose **Deformation**.

2 On the **Displacement** toolbar, click **Plot**.

Now create a plot of the shear stress in the local coordinate system of the piezoresistor, to compare with .

3D Plot Group 6

1 On 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 **Data set** 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 **Model>Component 1>Shell>Stress> Second Piola-Kirchhoff stress, local coordinate system>shell.Sl12 - Second Piola-Kirchhoff stress, local coordinate system, 12 component**.

In-Plane Shear Stress (Local Coordinates)

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

Max/Min Surface 1

1 On 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.Sl12 - Second Piola-Kirchhoff stress, local coordinate system, 12 component**.

3 On the **In-Plane Shear Stress (Local Coordinates)** toolbar, click **Plot**.

Create a line plot of the shear stress in the local coordinate system of the piezoresistor, to compare with .

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**, 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 **Model>Component 1>Shell>Stress>Second Piola-Kirchhoff stress, local coordinate system>shell.Sl12 - Second Piola-Kirchhoff stress, local coordinate system, 12 component**.

4 On 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 .

Selection

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

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

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

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

5 Click **Zoom to Selection**.

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

3D Plot Group 8

1 On 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 **Data set** 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, under **Results** 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 **Model>Component 1>Electric Currents, Shell>Currents and charge>ecs.tJx,...,ecs.tJz - Tangential current density**.

3 Locate the **Coloring and Style** section. Select the **Scale factor** check box.

4 In the associated text field, type `2e-9`.

5 In the **Number of arrows** text field, type `3000`.

6 From the **Color** list, choose **Blue**.

7 On the **Current and Voltage** toolbar, click **Plot**.

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

Global Evaluation 1

1 On 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 **Model>Component 1>Electric Currents, Shell>Terminals>ecs.I0_1 - Terminal current**.

3 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Electric Currents, Shell>Terminals>ecs.V0_2 - Terminal voltage**.

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**.

Created in COMSOL Multiphysics 5.3

Residual Stress in a Thin-Film Resonator—2D

Introduction

Almost all surface-micromachined thin films experience residual stress as a result of the fabrication process. The most common source of residual stress is thermal stress, which is caused by a change in temperature experienced during the fabrication sequence and also due to the difference in the coefficient of thermal expansion between the film and the substrate. This tutorial shows how to model thermal residual stress due to a temperature difference and how it changes the resonant frequency of a thin-film resonator. The substrate is not included in the model and it is also assumed that at a given state (which indicates a particular step of the process sequence), the temperature is uniform throughout the cantilever.

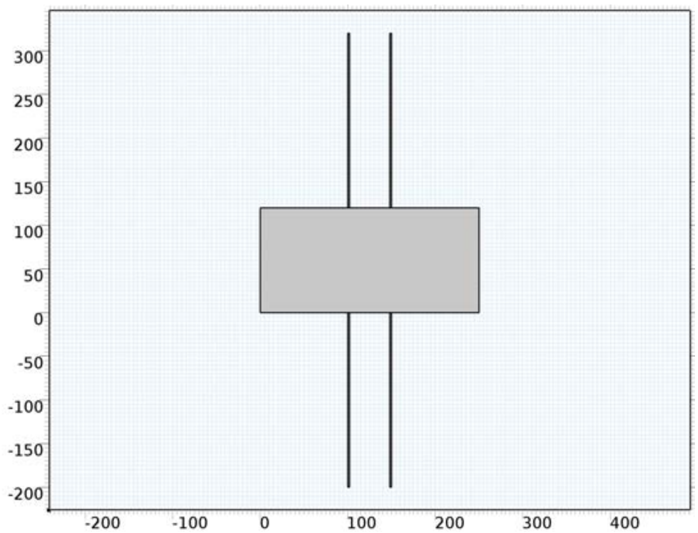


Figure 1: A thin-film resonator with four straight cantilever beam springs.

The tutorial investigates two design choices; a resonator with straight cantilevers (Figure 1) and another one with folded cantilevers (Figure 2). For each of the designs, the resonant frequency is computed for the cases when the structure is unstressed and when it is subjected to a residual thermal stress. The results obtained are compared with analytical solutions.

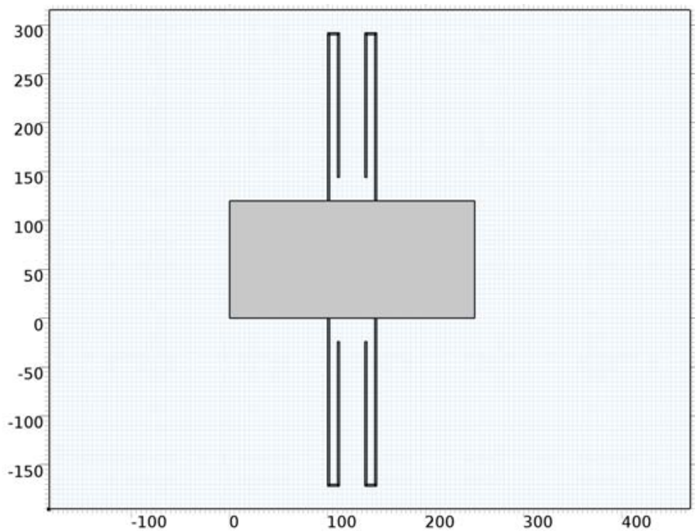


Figure 2: A thin-film resonator with four folded cantilever beam springs.

Model Definition

This tutorial uses the dimensions and material properties presented in Table 1 and Table 2. These values were obtained from the example in Chapter 27.2.5 in Ref. 1. It calculates the length of the folded cantilever using the equivalent spring-constant relationship discussed later.

This simulation models thermal residual stress using the Thermal Expansion feature in the Solid Mechanics interface. The coefficient of thermal expansion is computed by assuming a residual stress of 50 MPa in the straight cantilevers, a film deposition temperature of 605 °C (see Chapter 16.13.2.3 in Ref. 1) and a room temperature of 25 °C.

TABLE 1: DIMENSIONS OF THE STRUCTURE.

PARAMETER	STRAIGHT CANTILEVERS	FOLDED CANTILEVERS			PLATE
		L1	L2	L3	
Length	200 μm	170 μm	10 μm	146 μm	250 μm
Width	2 μm	2 μm	2 μm	2 μm	120 μm
Thickness	2.25 μm	2.25 μm	2.25 μm	2.25 μm	2.25 μm

TABLE 2: MATERIAL PROPERTIES OF THE STRUCTURE.

PROPERTY	VALUE
Material	polysilicon
Young's modulus	155 GPa
Poisson's ratio	0.23
Density	2330 kg/m^3
T_0	605 °C
T_I	25 °C

In order to determine the eigenfrequencies for the case with residual stress, a Prestressed-Eigenfrequency Study is used. This predefined study type first solves for a static thermal expansion problem to compute the residual stress. The solution of this static problem is then used to create a shift in the linearization point around which the eigenfrequencies are then computed. This approach accurately computes the shift in eigenfrequency by accounting for the stress-stiffening effect.

2D ANALYTICAL MODEL

For a lateral resonator with four cantilever-beam springs, the first in-plane bending resonant frequency is given by Equation 1.

$$f_0 \approx \frac{1}{2\pi}\sqrt{\frac{4Etb^3}{mL^3} + \frac{24\sigma_r tb}{5mL}} \tag{1}$$

Here m is the mass of the resonator plate, E is Young's modulus, L is the length of each cantilever arm, b is its width, t is its thickness, and σ_r is the residual stress in each cantilever. In this tutorial, the stress is assumed to be purely because of temperature difference but in reality it could be a sum of external stresses, the thermal stress, and intrinsic components. Assuming the material is isotropic, the stress is constant through the film thickness, and the stress component in the direction normal to the substrate is zero (i.e. plane stress). The stress-strain relationship is then given by Equation 2. Here ν is the Poisson's ratio.

$$\sigma_r = \left(\frac{E}{1-\nu}\right)\varepsilon \tag{2}$$

The strain comes from $\varepsilon = \alpha\Delta T$ where α is the thermal-expansion coefficient of the cantilever material and ΔT is the difference between the deposition temperature and the normal operating temperature.

Thermal residual stress in thin-film spring structures is typically relieved by folding the flexures as shown in Figure 2. The flexures relieve axial stress because each is free to expand or contract in the axial direction.

The basic folded structure is a U-shaped spring. For springs in series, the equivalent spring constant is given by Equation 3.

$$\frac{1}{k_{\mathrm{eq}}} = \frac{1}{k_1} + \frac{1}{k_2} + \frac{1}{k_3} \tag{3}$$

The first and third springs are cantilever beams. The equivalent spring constant for these can be computed as $k = 3EI/L^3$, where I is the moment of inertia. For a beam with rectangular cross-section and for a rotation about the local y-axis of the beam (the axis parallel to the in-plane width), the moment of inertia is $I = bt^3/12$, where b is the width and t is the structure's thickness. You can treat the second spring as a column with a spring constant of $k = AE/L$, where A is the cross-sectional area $A = bt$. Assuming the spring thickness (t) and width (b) are the same everywhere, Equation 3 can be used to find the equivalent length of each set of folded springs. The equivalent length can be expressed in terms of the out-of-plane thickness (t) and the length of each of the three sections of a folded cantilever as shown in Equation 4.

$$L_{\mathrm{eq}}^3 = L_1^3 + \frac{t^2 L_2}{4} + L_3^3, \tag{4}$$

Using the information provided in Table 1 and Table 2 and by using Equation 1 and Equation 2, one could compute the resonant frequency for the unstressed and stressed thin-film resonator with straight cantilever. Additionally by using Equation 4 along with the other information, one could also find the resonant frequency for the unstressed resonator with folded cantilevers. Note that the residual stress in the folded cantilevers are negligible by design and hence there is no need to compute the resonant frequency for this scenario. A summary of the analytical results are shown in Table 3 where they are compared with the solution obtained from the 2D plane-stress COMSOL model.

Results and Discussion

Table 3 summarizes the resonant frequencies for the first in-plane bending eigenmode. For the 2D COMSOL models this is the lowest (first) eigenmode. As the table shows, the resonant frequency for the straight cantilevers increases significantly when the model includes residual stress. The model results agree closely with the analytical estimates. As expected, the stress sensitivity of the resonant frequency is reduced by folding the cantilevers.

TABLE 3: RESONANT FREQUENCIES WITH AND WITHOUT RESIDUAL STRESS.

	STRAIGHT CANTILEVERS		FOLDED CANTILEVERS	
	ANALYTICAL	2D MODEL	ANALYTICAL	2D MODEL
Without stress	14.99 kHz	14.82 kHz	14.97 kHz	14.11 kHz
With residual stress	33.08 kHz	32.05 kHz	-	14.22 kHz

Figure 3 and Figure 4 show the first in-plane bending resonance mode for the unstressed resonator with straight and folded cantilevers respectively.

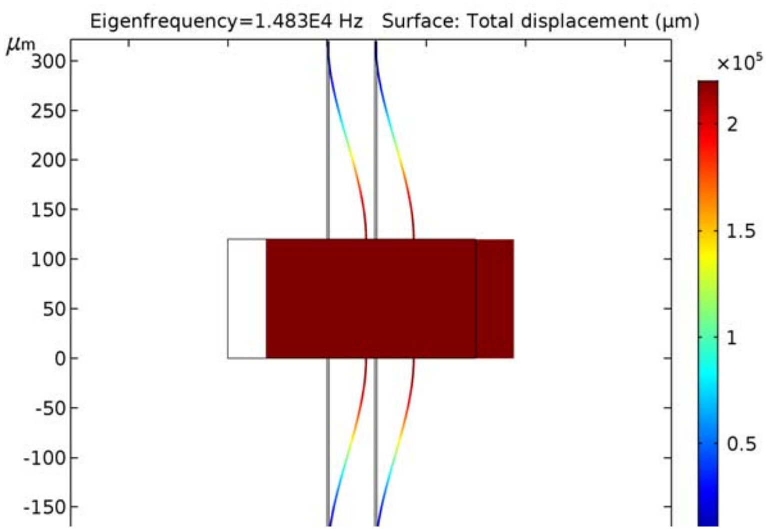


Figure 3: The first in-plane bending eigenmode of the unstressed resonator with straight cantilevers.

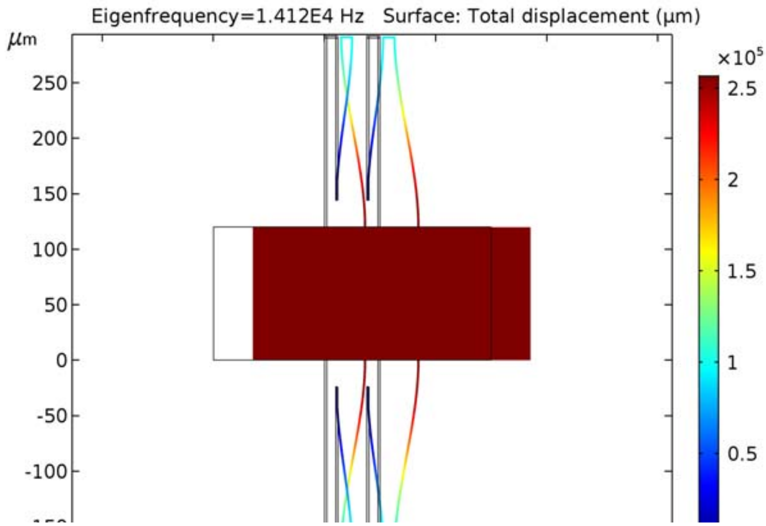


Figure 4: The first in-plane bending eigenmode of the unstressed resonator with folded cantilevers.

Figure 5 and Figure 6 show the first in-plane bending resonance mode for the resonator with straight and folded cantilevers respectively when they have a residual thermal stress.

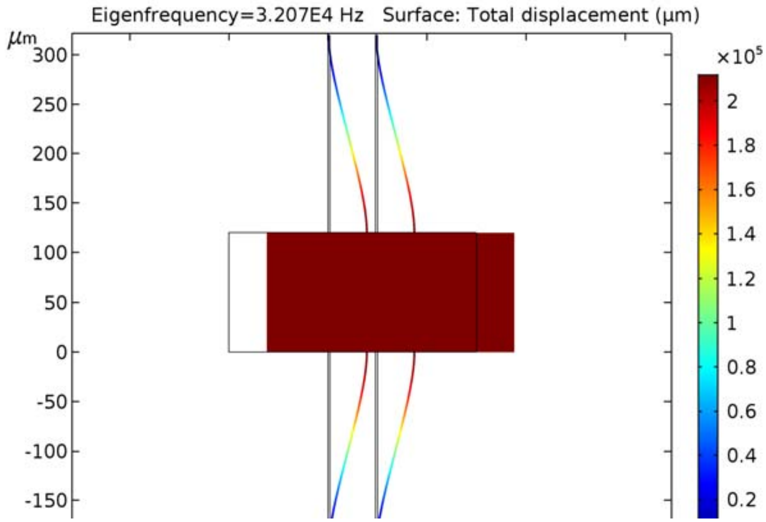


Figure 5: The first in-plane bending eigenmode of the resonator with straight cantilevers having residual thermal stress.

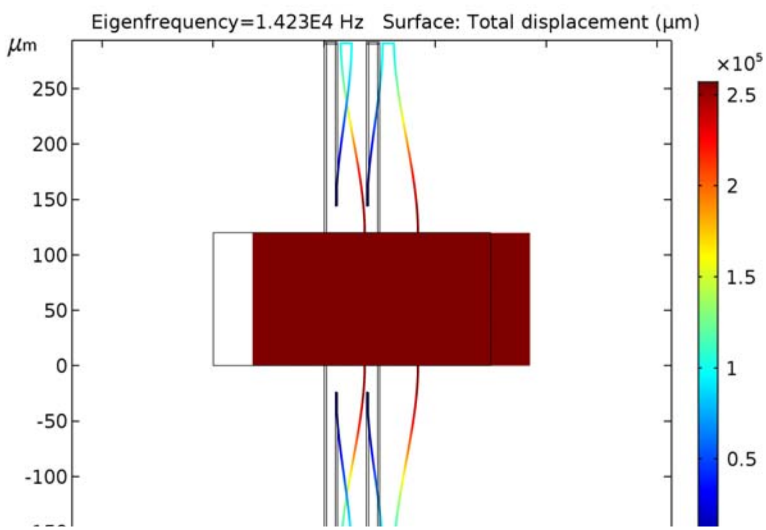


Figure 6: The first in-plane bending eigenmode of the resonator with folded cantilevers having residual thermal stress.

Figure 7 and Figure 8 show the residual thermal stress (von Mises stress) distribution in the resonator with straight and folded cantilevers respectively when they are cooled from 605 °C to 25 °C. Figure 7 shows that the residual stress is almost uniform in the straight cantilever and is about 49 MPa. The maximum stress is about 55 MPa at the two ends of the cantilevers. Figure 8 shows that the folded configuration significantly reduces the residual stress build-up. In this case the residual stress is around 2 MPa in most part of the cantilever except near the fixed end where it is close to 39 MPa.

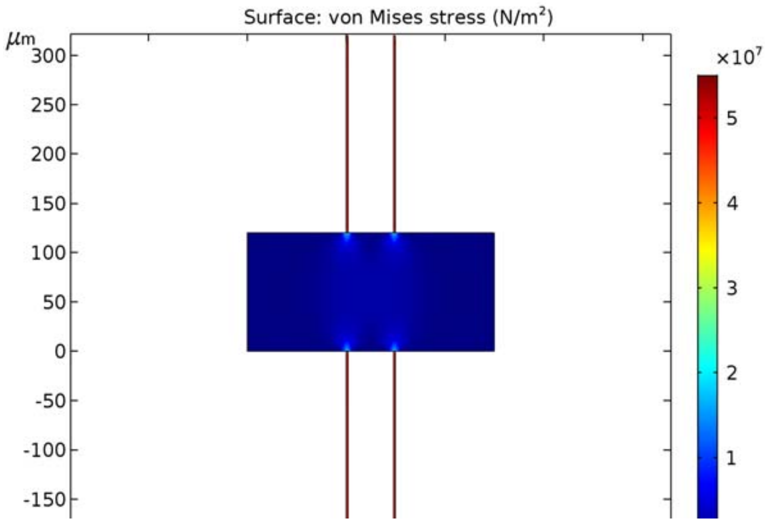


Figure 7: Residual thermal stress in the resonator with straight cantilevers when it is cooled from 605 *°C to* 25 *°C.*

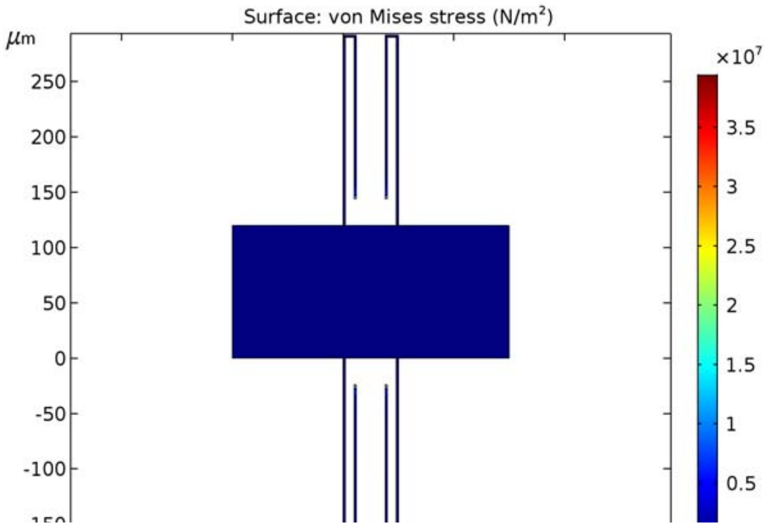


Figure 8: Residual thermal stress in the resonator with folded cantilevers when it is cooled from $605\ ^\circ C$ *to* $25\ ^\circ C$.

Reference

1. M. Gad-el-Hak, ed., *The MEMS Handbook*, CRC Press, London, 2002, ch. 16.12 and 27.2.5.

Application Library path: `MEMS_Module/Actuators/residual_stress_resonator_2d`

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 **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Eigenfrequency**.

6 Click **Done**.

Load in the required global parameters. As well as defining some model variables, these values are used later for comparison between the model and the analytical solution.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `residual_stress_resonator_2d_parameters.txt`.

First create a component to model the resonator with straight cantilevers. For convenience, the device geometry will be inserted from an existing file. You can read the instructions for creating the geometry in the .

GEOMETRY 1

1 On the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `residual_stress_resonator_2d_geom_sequence.mph`.

3 In the **Insert Sequence from File** dialog box, click **OK**.

4 On the **Geometry** toolbar, click **Build All**.

Next set up the required solid mechanics physics for the problem by adding a **Thermal Expansion** sub-feature and specifying the fixed boundaries.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.

3 From the list, choose **Plane stress**.

4 Locate the **Thickness** section. In the d text field, type `thickness`.

Thermal Expansion 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** right-click **Linear Elastic Material 1** and choose **Thermal Expansion**.
2. In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.
3. In the T text field, type `T0`.
4. Locate the **Thermal Expansion Properties** section. In the T_{ref} text field, type `T1`.

Fixed Constraint 1

1. In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.
2. In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.
3. Click **Paste Selection**.
4. In the **Paste Selection** dialog box, type `5,9,15,19` in the **Selection** text field.
5. Click **OK**.

Now add a new material to the component in order to define the required physical properties of the device.

MATERIALS

Material 1 (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, locate the **Material Contents** section.
3. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	`E1`	Pa	Basic
Poisson's ratio	nu	`nu1`	1	Basic
Density	rho	`rho1`	kg/m³	Basic
Coefficient of thermal expansion	alpha	`daT`	1/K	Basic

Configure a suitable mesh, a **Mapped** mesh is appropriate for this device geometry.

MESH 1

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `6,8,16,18` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Distribution**, locate the **Distribution** section.

8 In the **Number of elements** text field, type `2`.

Size

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

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

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

4 Click **Build All**.

Add a second component to model the resonator with folded cantilevers. As with the first component, the device geometry will be imported for convenience.

ROOT

On the **Home** toolbar, click **Component** and choose **Add Component>2D**.

GEOMETRY 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

2 On the **Geometry** toolbar, click **Insert Sequence**.

3 Browse to the model's Application Libraries folder and double-click the file `residual_stress_resonator_2d_geom_sequence.mph`.

4 In the **Insert Sequence from File** dialog box, select **Geometry 2** in the **Select geometry sequence to insert** list.

5 Click **OK**.

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

Now the solid mechanics physics can be configured, as with the first component. In addition, a material will be added to define the required properties of the second device and an appropriate mesh will be created.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2 Go to the **Add Physics** window.
3 In the tree, select **Structural Mechanics>Solid Mechanics (solid)**.
4 Click **Add to Component** in the window toolbar.
5 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

SOLID MECHANICS 2 (SOLID2)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Solid Mechanics 2 (solid2)**.
2 In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.
3 From the list, choose **Plane stress**.
4 Locate the **Thickness** section. In the d text field, type `thickness`.

Thermal Expansion 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Solid Mechanics 2 (solid2)** right-click **Linear Elastic Material 1** and choose **Thermal Expansion**.
2 In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.
3 In the T text field, type `T0`.
4 Locate the **Thermal Expansion Properties** section. In the T_{ref} text field, type `T1`.

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Solid Mechanics 2 (solid2)** and choose **Fixed Constraint**.
2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type `28,30,42,44` in the **Selection** text field.
5 Click **OK**.

MATERIALS

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	E1	Pa	Basic
Poisson's ratio	nu	nu1	1	Basic
Density	rho	rho1	kg/m³	Basic
Coefficient of thermal expansion	alpha	daT	1/K	Basic

MESH 2

Distribution 1

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Mesh 2** and choose **Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `4,8,10,11,28,30,38,42,44,45,60,62` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Distribution**, locate the **Distribution** section.

8 In the **Number of elements** text field, type `2`.

Size

1 In the **Model Builder** window, under **Component 2 (comp2)>Mesh 2** click **Size**.

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

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

4 Click **Build All**.

In order to perform the required computations four studies are required. The first two studies, one for each of the components, are for the case of zero-stress. These studies require one **Eigenfrequency** solver step, which will be used to calculate the eigenfrequency and mode of each resonator.

STUDY 1

1 In the **Model Builder** window, right-click **Study 1** and choose **Rename**.

2 In the **Rename Study** dialog box, type `Study 1 - Straight Cantilever, No Stress` in the **New label** text field.

3 Click **OK**.

STUDY 1 - STRAIGHT CANTILEVER, NO STRESS

Step 1: Eigenfrequency

1 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check box for the **Solid Mechanics 2 (solid2)** interface.

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> Eigenfrequency**.

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: Eigenfrequency

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

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the **Solid Mechanics (solid)** interface.

4 In the **Model Builder** window, right-click **Study 2** and choose **Rename**.

5 In the **Rename Study** dialog box, type `Study 2 - Folded Cantilever, No Stress` in the **New label** text field.

6 Click **OK**.

The second two studies require two study steps: an initial **Stationary** study step is used to calculate the residual thermal stress due to the difference between the fabrication and operation temperatures; the solution to this step is then used to shift the linerization point around which the eigenfrequenies are computed in a subsequent **Eigenfrequency** study step. **Prestressed Analysis, Eigenfrequency** studies are used as this study type contains the required study steps by default.

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> Prestressed Analysis, Eigenfrequency**.

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 **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the **Solid Mechanics 2 (solid2)** interface.

Step 2: Eigenfrequency

1 In the **Model Builder** window, under **Study 3** click **Step 2: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the **Solid Mechanics 2 (solid2)** interface.

4 In the **Model Builder** window, right-click **Study 3** and choose **Rename**.

5 In the **Rename Study** dialog box, type `Study 3 - Straight Cantilever, Residual Stress` in the **New label** text field.

6 Click **OK**.

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> Prestressed Analysis, Eigenfrequency**.

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

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

STUDY 4

Step 1: Stationary

1 In the **Model Builder** window, under **Study 4** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the **Solid Mechanics (solid)** interface.

Step 2: Eigenfrequency

1 In the **Model Builder** window, under **Study 4** click **Step 2: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the **Solid Mechanics (solid)** interface.

4 In the **Model Builder** window, right-click **Study 4** and choose **Rename**.

5 In the **Rename Study** dialog box, type `Study 4 - Folded Cantilever, Residual Stress` in the **New label** text field.

6 Click **OK**.

The studies can now be solved and the results visualized.

STUDY 1 - STRAIGHT CANTILEVER, NO STRESS

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

RESULTS

Mode Shape (solid)

1 In the **Model Builder** window, under **Results** right-click **Mode Shape (solid)** and choose **Rename**.

2 In the **Rename 2D Plot Group** dialog box, type `Straight Cantilever, No Stress` in the **New label** text field.

3 Click **OK**.

4 On the **Straight Cantilever, No Stress** toolbar, click **Plot**.

STUDY 2 - FOLDED CANTILEVER, NO STRESS

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

RESULTS

Mode Shape (solid2)

1 In the **Model Builder** window, under **Results** right-click **Mode Shape (solid2)** and choose **Rename**.

2 In the **Rename 2D Plot Group** dialog box, type `Folded Cantilever, No Stress` in the **New label** text field.

3 Click **OK**.

4 On the **Folded Cantilever, No Stress** toolbar, click **Plot**.

STUDY 3 - STRAIGHT CANTILEVER, RESIDUAL STRESS

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

RESULTS

Mode Shape (solid)

1 In the **Model Builder** window, under **Results** right-click **Mode Shape (solid)** and choose **Rename**.

2 In the **Rename 2D Plot Group** dialog box, type `Straight Cantilever, Residual Stress` in the **New label** text field.

3 Click **OK**.

4 On the **Straight Cantilever, Residual Stress** toolbar, click **Plot**.

STUDY 4 - FOLDED CANTILEVER, RESIDUAL STRESS

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

RESULTS

Mode Shape (solid2)

1 In the **Model Builder** window, under **Results** right-click **Mode Shape (solid2)** and choose **Rename**.

2 In the **Rename 2D Plot Group** dialog box, type `Folded Cantilever, Residual Stress` in the **New label** text field.

3 Click **OK**.

4 On the **Folded Cantilever, Residual Stress** toolbar, click **Plot**.

2D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Model Builder** window, right-click **2D Plot Group 5** and choose **Rename**.

3 In the **Rename 2D Plot Group** dialog box, type `Residual Stress in Straight Cantilever` in the **New label** text field.

4 Click **OK**.

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

6 From the **Data set** list, choose **Study 3 - Straight Cantilever, Residual Stress/ Solution Store 1 (7) (sol4)**.

Surface 1

1 Right-click **Residual Stress in Straight Cantilever** and choose **Surface**.

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

3 In the **Expression** text field, type `solid.mises`.

4 On the **Residual Stress in Straight Cantilever** toolbar, click **Plot**.

2D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Model Builder** window, right-click **2D Plot Group 6** and choose **Rename**.

3 In the **Rename 2D Plot Group** dialog box, type `Residual Stress in Folded Cantilever` in the **New label** text field.

4 Click **OK**.

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

6 From the **Data set** list, choose **Study 4 - Folded Cantilever, Residual Stress/ Solution Store 2 (12) (sol6)**.

Surface 1

1 Right-click **Residual Stress in Folded Cantilever** and choose **Surface**.

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

3 In the **Expression** text field, type `solid2.mises`.

4 On the **Residual Stress in Folded Cantilever** toolbar, click **Plot**.

Appendix — Geometry Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Done**.

GEOMETRY 1

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**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `250`.

4 In the **Height** text field, type `120`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

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

4 In the **Height** text field, type `200`.

5 Locate the **Position** section. In the **x** text field, type `100`.

6 In the **y** text field, type `120`.

Array 1 (arr1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **r2** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `2`.

5 In the **y size** text field, type `2`.

6 Locate the **Displacement** section. In the **x** text field, type `48`.

7 In the **y** text field, type `-320`.

8 Click **Build All Objects**.

ROOT

On the **Home** toolbar, click **Component** and choose **Add Component>2D**.

GEOMETRY 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

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

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

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 250.

4 In the **Height** text field, type 120.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

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

4 In the **Height** text field, type 172.

5 Locate the **Position** section. In the **x** text field, type 100.

6 In the **y** text field, type 120.

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 12.

4 In the **Height** text field, type 2.

5 Locate the **Position** section. In the **x** text field, type 100.

6 In the **y** text field, type 290.

Rectangle 4 (r4)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

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

4 In the **Height** text field, type 148.

5 Locate the **Position** section. In the **x** text field, type 110.

6 In the **y** text field, type 144.

Mirror 1 (mir1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

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

3 Select the objects **r3**, **r2**, and **r4** only.

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

5 Select the **Keep input objects** check box.

6 Locate the **Point on Line of Reflection** section. In the **x** text field, type `125`.

Mirror 2 (mir2)

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Select the objects **mir1(1)**, **r3**, **mir1(3)**, **r2**, **r4**, and **mir1(2)** only.

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

4 Select the **Keep input objects** check box.

5 Locate the **Point on Line of Reflection** section. In the **y** text field, type `60`.

6 Locate the **Normal Vector to Line of Reflection** section. In the **x** text field, type `0`.

7 In the **y** text field, type `1`.

8 Click **Build All Objects**.

Created in COMSOL Multiphysics 5.3

Residual Stress in a Thin-Film Resonator—3D

Introduction

Almost all surface-micromachined thin films experience residual stress as a result of the fabrication process. The most common source of residual stress is thermal stress, which is caused by a change in temperature experienced during the fabrication sequence and also due to the difference in the coefficient of thermal expansion between the film and the substrate. This tutorial shows how to model thermal residual stress due to a temperature difference and how it changes the resonant frequency of a thin-film resonator. The substrate is not included in the model and it is also assumed that at a given state (which indicates a particular step of the process sequence), the temperature is uniform throughout the cantilever.

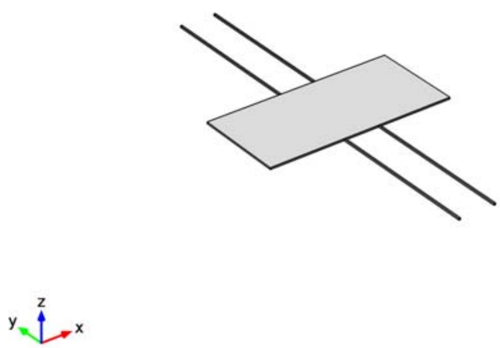


Figure 1: A thin-film resonator with four straight cantilever beam springs.

The tutorial investigates two design choices; a resonator with straight cantilevers (Figure 1) and another one with folded cantilevers (Figure 2). For each of the designs, the resonant frequency is computed for the cases when the structure is unstressed and when it is subjected to a residual thermal stress. The results obtained from these 3D models can be compared with analytical solutions by referring to the Application Libraries tutorial Residual Stress in a Thin-Film Resonator—2D.

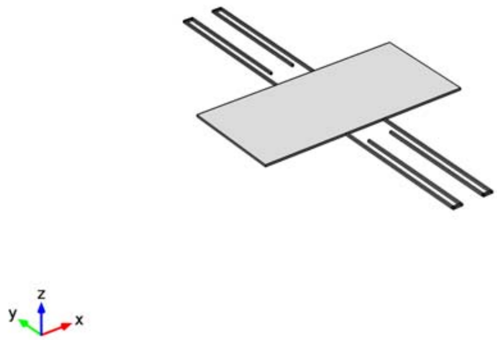


Figure 2: A thin-film resonator with four folded cantilever beam springs.

Model Definition

This tutorial uses the dimensions and material properties presented in Table 1 and Table 2. These values were obtained from the example in Chapter 27.2.5 in Ref. 1. This simulation models thermal residual stress using the Thermal Expansion feature in the Solid Mechanics interface. The coefficient of thermal expansion is computed by assuming a residual stress of 50 MPa in the straight cantilevers, a film deposition temperature of 605 °C (see Chapter 16.13.2.3 in Ref. 1) and a room temperature of 25 °C.

TABLE 1: DIMENSIONS OF THE STRUCTURE.

PARAMETER	STRAIGHT CANTILEVERS	FOLDED CANTILEVERS			PLATE
		L1	L2	L3	
Length	200 μm	170 μm	10 μm	146 μm	250 μm
Width	2 μm	2 μm	2 μm	2 μm	120 μm
Thickness	2.25 μm	2.25 μm	2.25 μm	2.25 μm	2.25 μm

TABLE 2: MATERIAL PROPERTIES OF THE STRUCTURE.

PROPERTY	VALUE
Material	polysilicon
Young's modulus	155 GPa
Poisson's ratio	0.23
Density	2330 kg/m^3
T_0	605 °C
T_l	25 °C

In order to determine the eigenfrequencies for the case with residual stress, a Prestressed-Eigenfrequency Study is used. This predefined study type first solves for a static thermal expansion problem to compute the residual stress. The solution of this static problem is then used to create a shift in the linearization point around which the eigenfrequencies are then computed. This approach accurately computes the shift in eigenfrequency by accounting for the stress-stiffening effect.

Results and Discussion

Table 3 summarizes the resonant frequencies for the first in-plane bending eigenmode. The solutions from the 3D models are compared with those obtained from a 2D plane-stress model available in the Application Libraries tutorial Residual Stress in a Thin-Film Resonator—2D. As the table shows, the resonant frequency for the straight cantilevers increases significantly when the model includes residual stress. As expected, the stress sensitivity of the resonant frequency is reduced by folding the cantilevers. The results from the 3D models agree closely with those obtained from the 2D models. This indicates that such thin-film resonators can be efficiently modeled using a 2D plane-stress approach.

TABLE 3: RESONANT FREQUENCIES WITH AND WITHOUT RESIDUAL STRESS.

	STRAIGHT CANTILEVERS		FOLDED CANTILEVERS	
	2D MODEL	3D MODEL	2D MODEL	3D MODEL
Without stress	14.82 kHz	14.91 kHz	14.11 kHz	14.18 kHz
With residual stress	32.05 kHz	32.25 kHz	14.22 kHz	14.29 kHz

Note that for the 2D models, the first (lowest) eigenmode is the in-plane bending mode. However for the 3D models, the first (lowest) eigenmode is an out-of-plane torsional mode that you see in the default plot when you solve the Eigenfrequency study. The desired in-plane bending mode is the second eigenmode in the list of computed solutions. Hence, the main advantage of the 3D model is in finding any eigenmode where the resonator deforms largely in the out-of-plane direction.

Figure 3 and Figure 4 show the first in-plane bending resonance mode for the unstressed resonator with straight and folded cantilevers respectively.

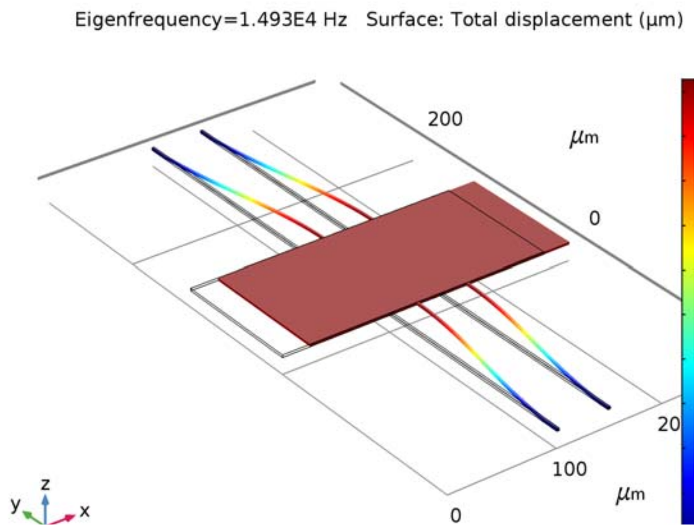


Figure 3: The first in-plane bending eigenmode of the unstressed resonator with straight cantilevers.

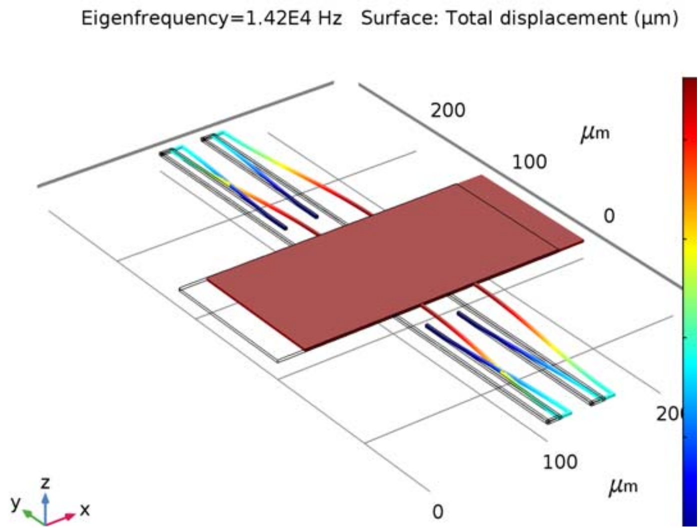


Figure 4: The first in-plane bending eigenmode of the unstressed resonator with folded cantilevers.

Figure 5 and Figure 7 show the first in-plane bending resonance mode for the resonator with straight and folded cantilevers respectively when they have a residual thermal stress.

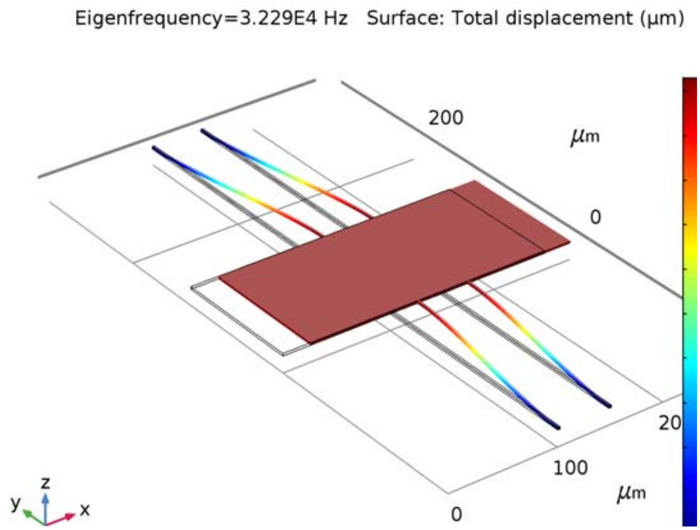


Figure 5: The first in-plane bending eigenmode of the resonator with straight cantilevers having residual thermal stress.

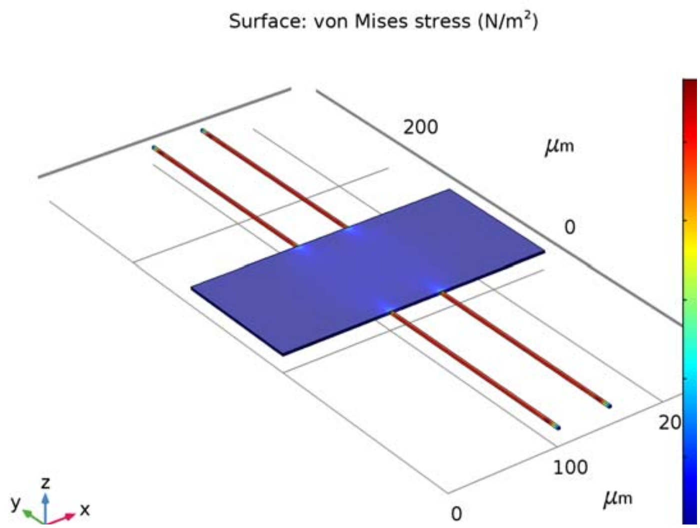


Figure 6: Residual thermal stress in the resonator with straight cantilevers when it is cooled from $605\ °C$ *to* $25\ °C$.

Figure 6 and Figure 8 show the residual thermal stress (von Mises stress) distribution in the resonator with straight and folded cantilevers respectively when they are cooled from 605 °C to 25 °C. Figure 6 shows that the residual stress is almost uniform in the straight cantilever and is about 49 MPa. Figure 8 shows that the folded configuration significantly reduces the residual stress build-up. In this case the residual stress is around 2 MPa in most part of the cantilever except near the fixed end where it is close to 44 MPa.

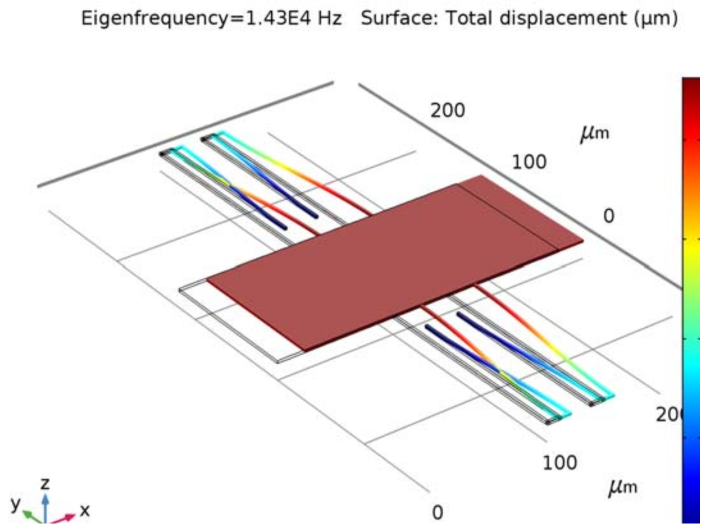


Figure 7: The first in-plane bending eigenmode of the resonator with folded cantilevers having residual thermal stress.

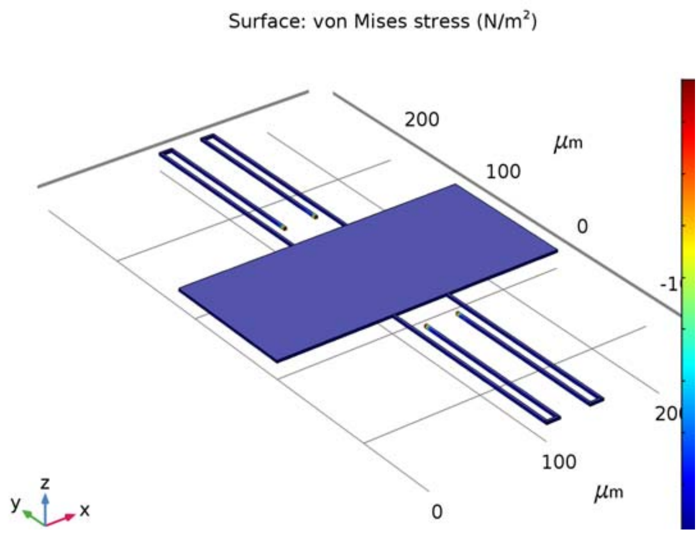


Figure 8: Residual thermal stress in the resonator with folded cantilevers when it is cooled from $605\ ^\circ C$ *to* $25\ ^\circ C$*.*

Reference

1. M. Gad-el-Hak, ed., *The MEMS Handbook*, CRC Press, London, 2002, ch. 16.12 and 27.2.5.

Application Library path: `MEMS_Module/Actuators/residual_stress_resonator_3d`

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>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Eigenfrequency**.

6 Click **Done**.

Load in the required global parameters. As well as defining some model variables, these values are used later for comparison between the model and the analytical solution.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `residual_stress_resonator_3d_parameters.txt`.

First create a component to model the resonator with straight cantilevers. For convenience, the device geometry will be inserted from an existing file. You can read the instructions for creating the geometry in the .

GEOMETRY 1

1 On the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `residual_stress_resonator_3d_geom_sequence.mph`.

3 In the **Insert Sequence from File** dialog box, click **OK**.

4 On the **Geometry** toolbar, click **Build All**.

Next set up the required solid mechanics physics for the problem by adding a **Thermal Expansion** sub-feature and specifying the fixed boundaries.

SOLID MECHANICS (SOLID)

Thermal Expansion 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** right-click **Linear Elastic Material 1** and choose **Thermal Expansion**.

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

3 In the T text field, type `T0`.

4 Locate the **Thermal Expansion Properties** section. In the T_{ref} text field, type `T1`.

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `7,15,21,29` in the **Selection** text field.

5 Click **OK**.

Now add a new material to the component in order to define the required physical properties of the device.

MATERIALS

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	E1	Pa	Basic
Poisson's ratio	nu	nu1	1	Basic
Density	rho	rho1	kg/m^3	Basic
Coefficient of thermal expansion	alpha	daT	1/K	Basic

Configure a suitable mesh, a **Mapped** and swept mesh is appropriate for this device geometry.

MESH 1

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Mapped**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `4,9,14,23,28` in the **Selection** text field.

5 Click **OK**.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `16,21,44,49` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Distribution**, locate the **Distribution** section.

7 In the **Number of elements** text field, type `2`.

Size

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

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

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

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 Right-click **Swept 1** and choose **Distribution**.

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

4 In the **Number of elements** text field, type `2`.

5 Click **Build All**.

Add a second component to model the resonator with folded cantilevers. As with the first component, the device geometry will be inserted for convenience.

ROOT

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

GEOMETRY 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

2 On the **Geometry** toolbar, click **Insert Sequence**.

3 Browse to the model's Application Libraries folder and double-click the file `residual_stress_resonator_3d_geom_sequence.mph`.

4 In the **Insert Sequence from File** dialog box, select **Geometry 2** in the **Select geometry sequence to insert** list.

5 Click **OK**.

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

Now the solid mechanics physics can be configured, as with the first component. In addition, a material will be added to define the required properties of the second device and an appropriate mesh will be created.

ADD PHYSICS

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

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

3 In the tree, select **Structural Mechanics>Solid Mechanics (solid)**.

4 Click **Add to Component** in the window toolbar.

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

SOLID MECHANICS 2 (SOLID2)

Thermal Expansion 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Solid Mechanics 2 (solid2)** right-click **Linear Elastic Material 1** and choose **Thermal Expansion**.

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

3 In the T text field, type `T0`.

4 Locate the **Thermal Expansion Properties** section. In the T_{ref} text field, type `T1`.

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Solid Mechanics 2 (solid2)** and choose **Fixed Constraint**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `46,48,68,70` in the **Selection** text field.

5 Click **OK**.

MATERIALS

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **Blank Material**.

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	E1	Pa	Basic
Poisson's ratio	nu	nu1	1	Basic
Density	rho	rho1	kg/m³	Basic
Coefficient of thermal expansion	alpha	daT	1/K	Basic

MESH 2

Mapped 1

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Mesh 2** and choose **More Operations>Mapped**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `4,9,13,18,22,27,36,41,45,50,54,63,67,72,76,81,88,93,97,102,106` in the **Selection** text field.

5 Click **OK**.

Distribution 1

1 Right-click **Component 2 (comp2)>Mesh 2>Mapped 1** and choose **Distribution**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `12,21,26,30,73,78,104,113,118,122,161,166` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Distribution**, locate the **Distribution** section.

7 In the **Number of elements** text field, type `2`.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 2** and choose **Swept**.

2 Right-click **Swept 1** and choose **Distribution**.

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

4 In the **Number of elements** text field, type `2`.

Size

1 In the **Model Builder** window, under **Component 2 (comp2)>Mesh 2** click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Predefined** list, choose **Extra fine**.
4 Click **Build All**.

In order to perform the required computations four studies are required. The first two studies, one for each of the components, are for the case of zero-stress. These studies require one **Eigenfrequency** solver step, which will be used to calculate the eigenfrequency and mode of each resonator.

STUDY 1

1 In the **Model Builder** window, right-click **Study 1** and choose **Rename**.
2 In the **Rename Study** dialog box, type `Study 1 - Straight Cantilever, No Stress` in the **New label** text field.
3 Click **OK**.

STUDY 1 - STRAIGHT CANTILEVER, NO STRESS

Step 1: Eigenfrequency

1 In the **Model Builder** window, expand the **Study 1 - Straight Cantilever, No Stress** node, then click **Step 1: Eigenfrequency**.
2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.
3 In the table, clear the **Solve for** check box for the **Solid Mechanics 2 (solid2)** interface.

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> Eigenfrequency**.
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: Eigenfrequency

1 In the **Model Builder** window, right-click **Study 2** and choose **Rename**.

2 In the **Rename Study** dialog box, type `Study 2 - Folded Cantilever, No Stress` in the **New label** text field.

3 Click **OK**.

STUDY 2 - FOLDED CANTILEVER, NO STRESS

Step 1: Eigenfrequency

1 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check box for the **Solid Mechanics (solid)** interface.

The second two studies require two study steps: an initial **Stationary** study step is used to calculate the residual thermal stress due to the difference between the fabrication and operation temperatures; the solution to this step is then used to shift the linerization point around which the eigenfrequenies are computed in a subsequent **Eigenfrequency** study step. **Prestressed Analysis, Eigenfrequency** studies are used as this study type contains the required study steps by default.

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> Prestressed Analysis, Eigenfrequency**.

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, right-click **Study 3** and choose **Rename**.

2 In the **Rename Study** dialog box, type `Study 3 - Straight Cantilever, Residual Stress` in the **New label** text field.

3 Click **OK**.

STUDY 3 - STRAIGHT CANTILEVER, RESIDUAL STRESS

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check box for the **Solid Mechanics 2 (solid2)** interface.

Step 2: Eigenfrequency

1 In the **Model Builder** window, under **Study 3 - Straight Cantilever, Residual Stress** click **Step 2: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the **Solid Mechanics 2 (solid2)** interface.

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> Prestressed Analysis, Eigenfrequency**.

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

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

STUDY 4

Step 1: Stationary

1 In the **Model Builder** window, right-click **Study 4** and choose **Rename**.

2 In the **Rename Study** dialog box, type `Study 4 - Folded Cantilever, Residual Stress` in the **New label** text field.

3 Click **OK**.

STUDY 4 - FOLDED CANTILEVER, RESIDUAL STRESS

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check box for the **Solid Mechanics (solid)** interface.

Step 2: Eigenfrequency

1 In the **Model Builder** window, under **Study 4 - Folded Cantilever, Residual Stress** click **Step 2: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the **Solid Mechanics (solid)** interface.

The studies can now be solved and the results visualized.

STUDY 1 - STRAIGHT CANTILEVER, NO STRESS

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

RESULTS

Mode Shape (solid)

1 In the **Model Builder** window, under **Results** right-click **Mode Shape (solid)** and choose **Rename**.
2 In the **Rename 3D Plot Group** dialog box, type `Straight Cantilever, No Stress` in the **New label** text field.
3 Click **OK**.
4 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
5 From the **Eigenfrequency (Hz)** list, choose **1.493E4**.
6 On the **Straight Cantilever, No Stress** toolbar, click **Plot**.

STUDY 2 - FOLDED CANTILEVER, NO STRESS

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

RESULTS

Mode Shape (solid2)

1 In the **Model Builder** window, under **Results** right-click **Mode Shape (solid2)** and choose **Rename**.
2 In the **Rename 3D Plot Group** dialog box, type `Folded Cantilever, No Stress` in the **New label** text field.
3 Click **OK**.
4 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
5 From the **Eigenfrequency (Hz)** list, choose **1.42E4**.
6 On the **Folded Cantilever, No Stress** toolbar, click **Plot**.

STUDY 3 - STRAIGHT CANTILEVER, RESIDUAL STRESS

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

RESULTS

Mode Shape (solid)

1 In the **Model Builder** window, under **Results** right-click **Mode Shape (solid)** and choose **Rename**.

2 In the **Rename 3D Plot Group** dialog box, type `Straight Cantilever, Residual Stress` in the **New label** text field.

3 Click **OK**.

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

5 From the **Eigenfrequency (Hz)** list, choose **3.229E4**.

6 On the **Straight Cantilever, Residual Stress** toolbar, click **Plot**.

STUDY 4 - FOLDED CANTILEVER, RESIDUAL STRESS

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

RESULTS

Mode Shape (solid2)

1 In the **Model Builder** window, under **Results** right-click **Mode Shape (solid2)** and choose **Rename**.

2 In the **Rename 3D Plot Group** dialog box, type `Folded Cantilever, Residual Stress` in the **New label** text field.

3 Click **OK**.

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

5 From the **Eigenfrequency (Hz)** list, choose **1.43E4**.

6 On the **Folded Cantilever, Residual Stress** toolbar, click **Plot**.

3D Plot Group 5

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

2 In the **Model Builder** window, right-click **3D Plot Group 5** and choose **Rename**.

3 In the **Rename 3D Plot Group** dialog box, type `Residual Stress in Straight Cantilever` in the **New label** text field.

4 Click **OK**.

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

6 From the **Data set** list, choose **Study 3 - Straight Cantilever, Residual Stress/ Solution Store 1 (7) (sol4)**.

Surface 1

1 Right-click **Residual Stress in Straight Cantilever** and choose **Surface**.

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

3 In the **Expression** text field, type `solid.mises`.

4 On the **Residual Stress in Straight Cantilever** toolbar, click **Plot**.

3D Plot Group 6

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

2 In the **Model Builder** window, right-click **3D Plot Group 6** and choose **Rename**.

3 In the **Rename 3D Plot Group** dialog box, type `Residual Stress in Folded Cantilever` in the **New label** text field.

4 Click **OK**.

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

6 From the **Data set** list, choose **Study 4 - Folded Cantilever, Residual Stress/ Solution Store 2 (12) (sol6)**.

Surface 1

1 Right-click **Residual Stress in Folded Cantilever** and choose **Surface**.

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

3 In the **Expression** text field, type `solid2.mises`.

4 On the **Residual Stress in Folded Cantilever** toolbar, click **Plot**.

Appendix — Geometry 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>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Done**.

GEOMETRY 1

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**.

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 250.

4 In the **Height** text field, type 120.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

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

4 In the **Height** text field, type 200.

5 Locate the **Position** section. In the **xw** text field, type 100.

6 In the **yw** text field, type 120.

Array 1 (arr1)

1 On the **Work Plane** toolbar, click **Transforms** and choose **Array**.

2 Select the object **r2** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **xw size** text field, type 2.

5 In the **yw size** text field, type 2.

6 Locate the **Displacement** section. In the **xw** text field, type 48.

7 In the **yw** text field, type -320.

Plane Geometry

1 On the **Work Plane** toolbar, click **Build All**.

2 In the **Model Builder** window, click **Geometry 1**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

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

3 In the table, enter the following settings:

Distances (µm)
2.25

4 Click **Build All Objects**.

ROOT

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

GEOMETRY 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

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

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

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 250.

4 In the **Height** text field, type 120.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

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

4 In the **Height** text field, type 172.

5 Locate the **Position** section. In the **xw** text field, type 100.

6 In the **yw** text field, type 120.

Rectangle 3 (r3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 12.

4 In the **Height** text field, type 2.

5 Locate the **Position** section. In the **xw** text field, type `100`.

6 In the **yw** text field, type `290`.

Rectangle 4 (r4)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

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

4 In the **Height** text field, type `148`.

5 Locate the **Position** section. In the **xw** text field, type `110`.

6 In the **yw** text field, type `144`.

Mirror 1 (mir1)

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

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

3 Select the objects **r3**, **r4**, and **r2** only.

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

5 Select the **Keep input objects** check box.

6 Locate the **Point on Line of Reflection** section. In the **xw** text field, type `125`.

Mirror 2 (mir2)

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

2 Select the objects **mir1(3)**, **r3**, **mir1(2)**, **mir1(1)**, **r4**, and **r2** only.

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

4 Select the **Keep input objects** check box.

5 Locate the **Point on Line of Reflection** section. In the **yw** text field, type `60`.

6 Locate the **Normal Vector to Line of Reflection** section. In the **xw** text field, type `0`.

7 In the **yw** text field, type `1`.

8 In the **Model Builder** window, click **Geometry 2**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

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

3 In the table, enter the following settings:

Distances (µm)
2.25

4 Click **Build All Objects**.

Created in COMSOL Multiphysics 5.3

Pull-in of an RF MEMS Switch

Introduction

This model analyzes an RF MEMS switch consisting of a thin micromechanical bridge suspended over a dielectric layer. A DC voltage greater than the pull-in voltage is applied across the switch, causing the bridge to collapse onto the dielectric layer with a resulting increase in the capacitance of the device. A penalty based contact force is implemented to model the contact forces as the bridge comes into contact with the dielectric.

Model Definition

Figure 1 shows the device geometry. The switch consists of a square polysilicon plate suspended 0.9 μm above a 0.1 μm thick thin film of Silicon Nitride (relative dielectric constant 7.5). Beneath the substrate is a silicon counter-electrode that is grounded. The plate is structurally anchored to the substrate by four rectangular flexures at its corners but is electrically isolated from it. Initially a small potential of just 1 mV is applied to the polysilicon. This voltage is sufficient to measure the DC capacitance of the device. After 25 μs the applied voltage is increased by 5 V with a step function that has a rise time of 10 μs. The applied voltage is greater than the pull-in voltage of the structure and the switch pulls down onto the nitride. This process results in an abrupt and significant change in the capacitance of the device. Due to the symmetry of the device, it is possible to model only a single quadrant of the structure. The electromechanics interface explicitly meshes the gap between the polysilicon and the nitride. Since it is not possible to collapse the mesh to zero thickness the mesh is compressed into the nitride layer as the structure deforms. The nitride layer itself is consequently not represented explicitly within the geometry, but instead is represented by means of a spatially varying function for the dielectric constant within the gap. The dielectric constant in the gap is represented by a smoothed step function. The mid point of the step is chosen to be slightly above the height of the dielectric, so that when the polysilicon is in contact with the nitride the dielectric constant in the gap takes the value of the nitride dielectric constant for throughout the gap.

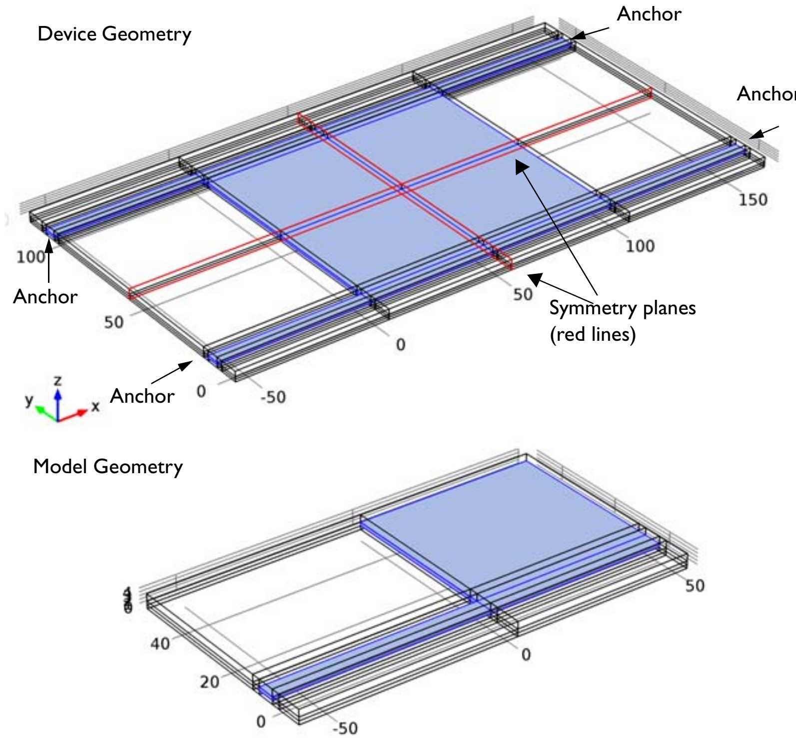


Figure 1: Top: Device geometry showing anchor points and symmetry planes. Bottom: Model geometry. Due to symmetry only one quadrant of the device needs to be modeled.

The contact between the polysilicon and the nitride is handled by an approximate penalty or barrier method, as described in Ref. 1. Stiff, nonlinear springs are used to represent the surface of the nitride. When the polysilicon is away from the nitride surface these springs have low stiffness and consequently have a negligible influence on the deformation. As the gap is reduced and approaches a predefined distance the spring becomes much stiffer and resists further closure. The contact forces $\boldsymbol{F}_c$ are given by:

$$\boldsymbol{F}_c = t_n - e_n \cdot g \qquad g < 0$$

$$\boldsymbol{F}_c = t_n + \exp\left(-\frac{e_n}{t_n} \cdot g\right) \qquad g \geq 0$$

where t_{n} is the input estimate of the contact force, e_n is the penalty stiffness, g is the gap, that is, the distance between the polysilicon and the nitride. Note that when this method is employed it is important to correctly tune the elastic stiffness and the contact force. The technique is an approximate one and does not correctly reproduce the details of the dynamics of contact. However the model is primarily concerned with estimating the time the switch takes to make contact and with computing the initial and final capacitance of the switch.

Results and Discussion

Figure 2 shows the spatial dependence of the total displacement when the device is pulled in. Most of the structure is in contact with the nitride and the bending occurs primarily in the flexures and in the vicinity of their attachment points. The form of the contact pressure, shown in Figure 3, is consistent with this observation and it is interesting to note that the largest forces occur in the vicinity of the flexures.

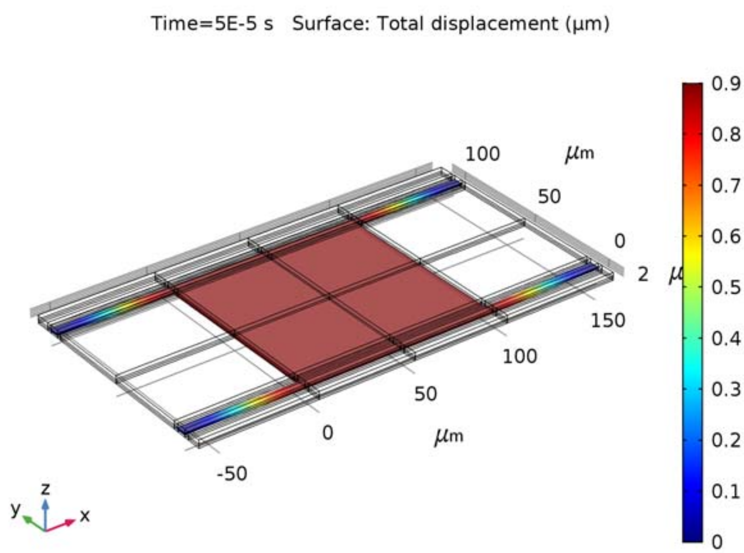


Figure 2: Displacement of the polysilicon when pulled in. Most of the polysilicon structure is in contact with the silicon nitride with a displacement of 0.9 µ*m.*

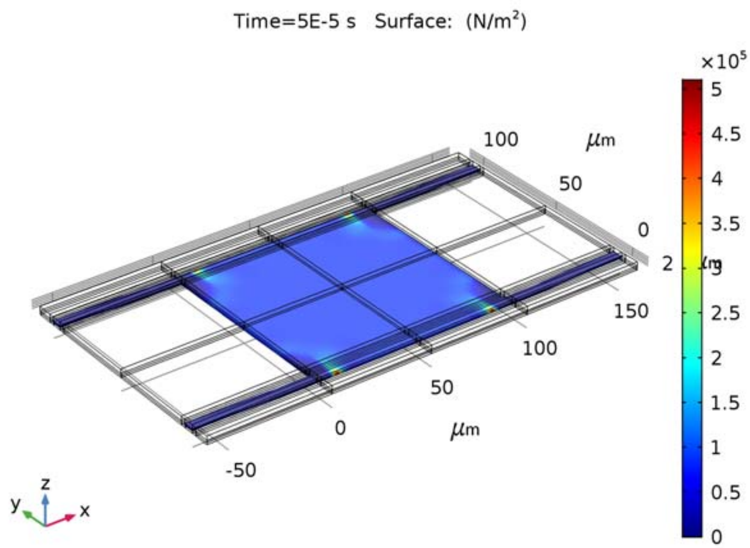


Figure 3: Contact forces acting on the polysilicon when the structure is pulled in.

Figure 4 shows the displacement of the switch as a function of time. The switch takes significantly longer to close than the timescale on which the voltage changes, primarily due to its inertia. Figure 5 shows the capacitance of the device as a function of time. The capacitance increases by a factor of approximately 55. Note that the capacitance changes on a significantly shorter timescale than the displacement.

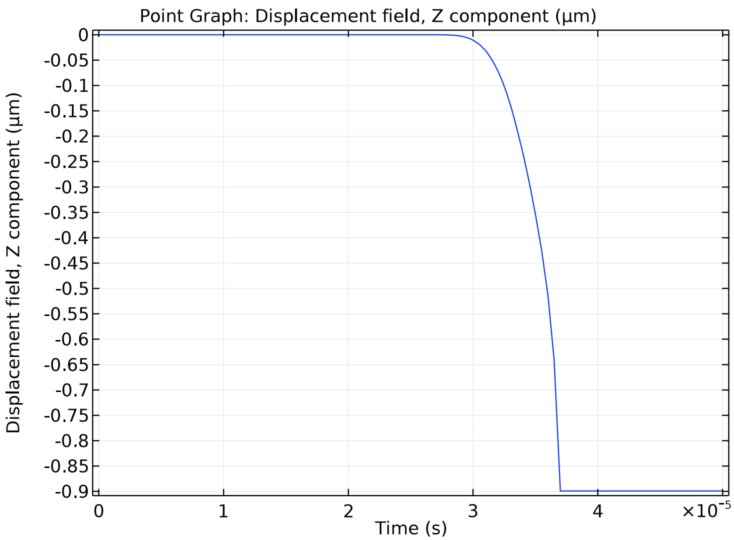


Figure 4: Displacement of the center of the device as a function of time.

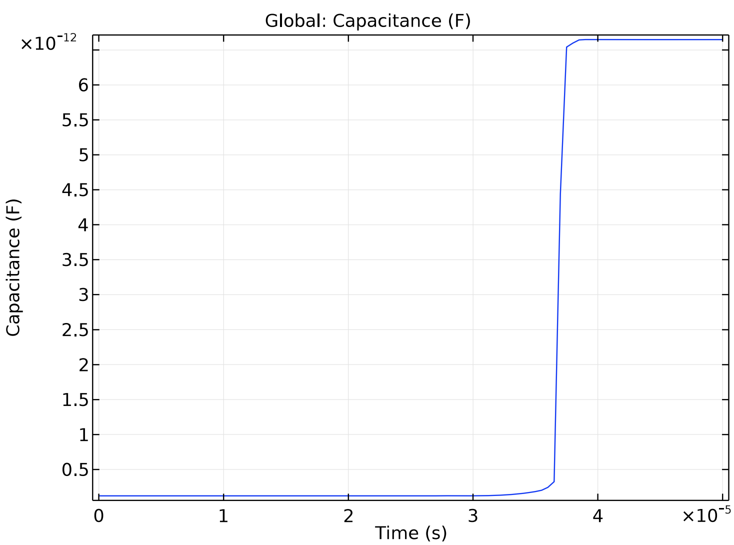


Figure 5: Capacitance of the device as a function of time. The transients in this plot that occur after the point of contact are not physical. The capacitance of the structure changes from 0.1 pF to 1.5 pF as a result of the pull-in.

Reference

1. Crisfield M. A., *Non-linear Finite Element Analysis of Solids and Structures, volume 2: Advanced Topics*, John Wiley & Sons Ltd., England, 1997.

Application Library path: `MEMS_Module/Actuators/rf_mems_switch`

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

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**.

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:

Name	Expression	Value	Description
V0	1[mV]	0.001 V	Initial voltage
Vstep	5[V]	5 V	Voltage step
insheight	100[nm]	1E-7 m	Insulator height
airheight	900[nm]	9E-7 m	Air height
en	1e15[Pa/m]	1E15 N/m³	Spring stiffness
tn	5e5[Pa]	5E5 Pa	Contact force

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
gap	airheight+w	m	
contactpressure	(gap<=0)*(tn-en*gap)+(gap>0)*tn*exp(-gap*en/tn)	N/m²	
Va	V0+Vstep*step2(t/1[s])	V	

GEOMETRY 1

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 110.

4 In the **Height** text field, type 5.

5 Locate the **Position** section. In the **xw** text field, type -60.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `50`.

4 In the **Height** text field, type `60`.

5 Locate the **Position** section. In the **yw** text field, type `-10`.

Rectangle 3 (r3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `110`.

4 In the **Height** text field, type `60`.

5 Locate the **Position** section. In the **xw** text field, type `-60`.

6 In the **yw** text field, type `-10`.

Rectangle 4 (r4)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `110`.

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

5 Locate the **Position** section. In the **xw** text field, type `-60`.

6 In the **yw** text field, type `-2.5`.

7 In the **Model Builder** window, click **Geometry 1**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

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

3 In the table, enter the following settings:

Distances (µm)
1
2
4

4 Right-click **Extrude 1 (ext1)** and choose **Build Selected**.

Form Union (fin)

1 On the **Geometry** toolbar, click **Build All**.

2 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

DEFINITIONS

Step 1 (step1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Step**.

2 In the **Settings** window for **Step**, locate the **Parameters** section.

3 In the **Location** text field, type `1.05*insheight`.

4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `0.05*insheight`.

Step 2 (step2)

1 On the **Home** toolbar, click **Functions** and choose **Local>Step**.

2 In the **Settings** window for **Step**, locate the **Parameters** section.

3 In the **Location** text field, type `3e-5`.

4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `1e-5`.

Define selections.

Explicit 1

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

2 In the **Model Builder** window, right-click **Explicit 1** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `Bridge` in the **New label** text field.

4 Click **OK**.

5 Select Domains 8, 23, 26, and 29 only.

Box 1

1 On the **Definitions** toolbar, click **Box**.

2 In the **Model Builder** window, right-click **Box 1** and choose **Rename**.

3 In the **Rename Box** dialog box, type `Gap` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Box**, locate the **Box Limits** section.

6 In the **z maximum** text field, type `1.1`.

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

Adjacent 1

1 On the **Definitions** toolbar, click **Adjacent**.

2 In the **Model Builder** window, right-click **Adjacent 1** and choose **Rename**.

3 In the **Rename Adjacent** dialog box, type `Bridge surface` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Adjacent**, locate the **Input Entities** section.

6 Under **Input selections**, click **Add**.

7 In the **Add** dialog box, select **Bridge** in the **Input selections** list.

8 Click **OK**.

Box 2

1 On the **Definitions** toolbar, click **Box**.

2 In the **Model Builder** window, right-click **Box 2** and choose **Rename**.

3 In the **Rename Box** dialog box, type `Base` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Box**, locate the **Geometric Entity Level** section.

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

7 Locate the **Box Limits** section. In the **z maximum** text field, type `0.1`.

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

Box 3

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 **z minimum** text field, type `0.9`.

5 In the **z maximum** text field, type `1.1`.

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

Intersection 1

1 On the **Definitions** toolbar, click **Intersection**.

2 In the **Model Builder** window, right-click **Intersection 1** and choose **Rename**.

3 In the **Rename Intersection** dialog box, type `Bridge lower side` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Intersection**, locate the **Geometric Entity Level** section.

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

7 Locate the **Input Entities** section. Under **Selections to intersect**, click **Add**.

8 In the **Add** dialog box, In the **Selections to intersect** list, choose **Bridge surface** and **Box 3**.

9 Click **OK**.

Box 4

1 On the **Definitions** toolbar, click **Box**.

2 In the **Model Builder** window, right-click **Box 4** and choose **Rename**.

3 In the **Rename Box** dialog box, type `Symmetry x` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Box**, locate the **Geometric Entity Level** section.

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

7 Locate the **Box Limits** section. In the **x minimum** text field, type `45`.

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

Box 5

1 On the **Definitions** toolbar, click **Box**.

2 In the **Model Builder** window, right-click **Box 5** and choose **Rename**.

3 In the **Rename Box** dialog box, type `Symmetry y` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Box**, locate the **Geometric Entity Level** section.

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

7 Locate the **Box Limits** section. In the **y minimum** text field, type `45`.

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

Load the materials.

ADD MATERIAL

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

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

3 In the tree, select **Built-In>Air**.

4 Click **Add to Component** in the window toolbar.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **MEMS>Semiconductors>Si - Polycrystalline Silicon**.

3 Click **Add to Component** in the window toolbar.

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Si - Polycrystalline Silicon (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Si - Polycrystalline Silicon (mat2)**.

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

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

Material 3 (mat3)

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

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

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

4 Click to expand the **Material properties** section. Locate the **Material Properties** section. From the **Material type** list, choose **Nonsolid**.

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

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	`7.5-step1(z)*6.5`	1	Basic

ELECTROMECHANICS (EMI)

In the **Model Builder** window, expand the **Electromechanics (emi)** node.

Linear Elastic Material 1

1 Right-click **Component 1 (comp1)>Electromechanics (emi)** 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 **Bridge**.

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Electromechanics (emi)** and choose the boundary condition **Structural>Fixed Constraint**.

2 Select Boundary 24 only.

Symmetry 1

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Structural> Symmetry**.

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

3 From the **Selection** list, choose **Symmetry x**.

Symmetry 2

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Structural> Symmetry**.

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

3 From the **Selection** list, choose **Symmetry y**.

Boundary Load 1

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Structural> Boundary Load**.

2 Select Boundaries 26, 79, 89, and 99 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	x
0	y
contactpressure	z

Prescribed Mesh Displacement 2

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Deformed Mesh> Prescribed Mesh Displacement**.

2 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry x**.

4 Locate the **Prescribed Mesh Displacement** section. Clear the **Prescribed y displacement** check box.

5 Clear the **Prescribed z displacement** check box.

Prescribed Mesh Displacement 3

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Deformed Mesh>Prescribed Mesh Displacement**.

2 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry y**.

4 Locate the **Prescribed Mesh Displacement** section. Clear the **Prescribed x displacement** check box.

5 Clear the **Prescribed z displacement** check box.

6 Right-click **Electromechanics (emi)** and choose the boundary condition **Electrical>Terminal**.

Define the voltage applied to the bridge lower side.

Terminal 1

1 In the **Settings** window for **Terminal**, locate the **Boundary Selection** section.

2 From the **Selection** list, choose **Bridge surface**.

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

4 In the V_0 text field, type `Va`.

5 Right-click **Electromechanics (emi)** and choose the boundary condition **Electrical>Terminal**.

Define the voltage applied to the base.

Terminal 2

1 In the **Settings** window for **Terminal**, locate the **Boundary Selection** section.

2 From the **Selection** list, choose **Base**.

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

4 In the V_0 text field, type `0`.

5 Right-click **Electromechanics (emi)** and choose the boundary condition **Electrical>Terminal**.

Add another **Terminal** to the bridge lower side. This terminal is used for the first (**Stationary**) study step. It will be disabled in the transient step which will make **Terminal 1** active.

Terminal 3

1 In the **Settings** window for **Terminal**, locate the **Boundary Selection** section.

2 From the **Selection** list, choose **Bridge surface**.

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

4 In the V_0 text field, type `V0`.

MESH 1

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Mapped**.

2 Select Boundaries 10, 20, 30, 40, 50, 63, 73, 83, 93, and 103 only.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 Right-click **Swept 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.

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

5 Locate the **Distribution** section. In the **Number of elements** text field, type `8`.

6 Click **Build All**.

Define the study steps. The first step (**Stationary**) is used to define the initial conditions of the transient problem (**Step 2**).

STUDY 1

Step 2: Time Dependent

1 On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Times** text field, type `range(0,5e-7,5e-5)`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Electromechanics (emi)>Terminal 3**.

6 Click **Disable**.

Change the default solver to a **Fully CoupledDirect** solver and use **BDF** time stepping.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

4 Locate the **Time Stepping** section. From the **Method** list, choose **BDF**.

5 From the **Steps taken by solver** list, choose **Intermediate**.

6 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** and choose **Fully Coupled**.

7 In the **Settings** window for **Fully Coupled**, click to expand the **Method and termination** section.

8 Locate the **Method and Termination** section. From the **Nonlinear method** list, choose **Automatic (Newton)**.

9 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** right-click **Direct** and choose **Enable**.

10 In the **Settings** window for **Direct**, locate the **General** section.

11 From the **Solver** list, choose **PARDISO**.

12 Click **Compute**.

Create the mirror solutions.

RESULTS

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 **Plane Data** section.

3 In the **x-coordinate** text field, type 50.

Mirror 3D 2

1 Right-click **Mirror 3D 1** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Mirror 3D 1**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **xz-planes**.

5 In the **y-coordinate** text field, type 50.

Displacement (emi)

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

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

3 From the **Data set** list, choose **Mirror 3D 2**.

4 On the **Displacement (emi)** toolbar, click **Plot**.

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

Potential (emi)

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

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

3 From the **Data set** list, choose **Mirror 3D 2**.

4 On the **Potential (emi)** toolbar, click **Plot**.

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

Displacement (emi) 1

1 In the **Model Builder** window, under **Results** right-click **Displacement (emi)** and choose **Duplicate**.

2 Right-click **Displacement (emi) 1** and choose **Rename**.

3 In the **Rename 3D Plot Group** dialog box, type `Contact force (emi)` in the **New label** text field.

4 Click **OK**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Contact force (emi)** node, then click **Surface 1**.

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

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

4 On the **Contact force (emi)** toolbar, click **Plot**.

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

1D Plot Group 4

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

2 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Rename**.

3 In the **Rename 1D Plot Group** dialog box, type `Displacement` in the **New label** text field.

4 Click **OK**.

Point Graph 1

1 Right-click **Displacement** and choose **Point Graph**.

2 Select Point 70 only.

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

4 In the **Expression** text field, type `w`.

5 On the **Displacement** toolbar, click **Plot**.

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

1D Plot Group 5

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

2 In the **Model Builder** window, right-click **1D Plot Group 5** and choose **Rename**.

3 In the **Rename 1D Plot Group** dialog box, type `Capacitance` in the **New label** text field.

4 Click **OK**.

Global 1

1 Right-click **Capacitance** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`4*emi.Q0_1/emi.V0_1`	`F`	`Capacitance`

4 Click to expand the **Legends** section. Clear the **Show legends** check box.

5 On the **Capacitance** toolbar, click **Plot**.

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

Surface Acoustic Wave Gas Sensor

Introduction

A surface acoustic wave (SAW) is an acoustic wave propagating along the surface of a solid material. Its amplitude decays rapidly, often exponentially, with the depth of the material. SAWs are featured in many kinds of electronic components, including filters, oscillators, and sensors. SAW devices typically use electrodes on a piezoelectric material to convert an electric signal to a SAW, and back again.

In this model, you investigate the resonance frequencies of a SAW gas sensor. The sensor consists of an interdigitated transducer (IDT) etched onto a piezoelectric $LiNbO_3$ (lithium niobate) substrate and covered with a thin polyisobutylene (PIB) film. The mass of the PIB film increases as PIB selectively adsorbs CH_2Cl_2 (dichloromethane, DCM) from air. This causes a shift in resonance which slightly lowers the resonance frequency for the same SAW mode.

Model Definition

Figure 1 shows a conceptual view of the gas sensor in this model.

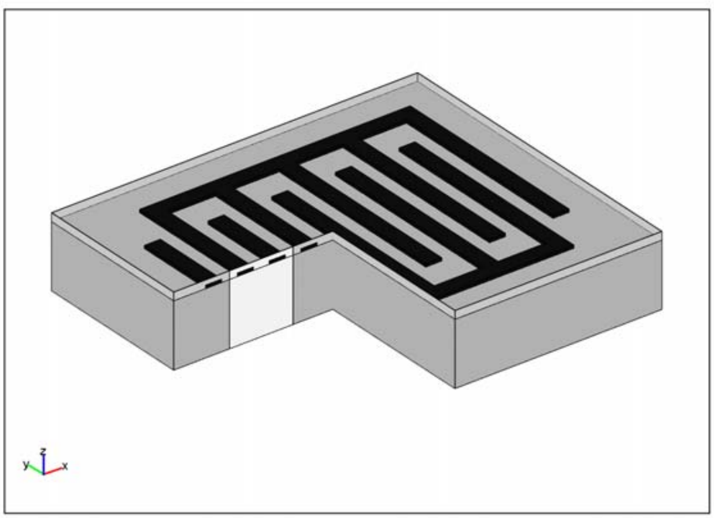


Figure 1: SAW gas sensor, showing the IDT electrodes (in black), the thin PIB film (light gray), and the $LiNbO_3$ substrate (dark gray). For the sake of clarity, the dimensions are not to scale and the IDT has fewer electrodes than in common devices. A slice of the geometry is removed to reveal the modeled unit cell (in white).

IDTs used in SAW devices may have hundreds of identical electrodes, and each electrode can be about 100 times longer than it is wide. You can therefore neglect the edge effects and reduce the model geometry to the periodic unit cell shown in Figure 2. The height of this cell does not have to extend all the way to the bottom of the substrate but only a few wavelengths down, so that the SAW has almost died out at the lower boundary.

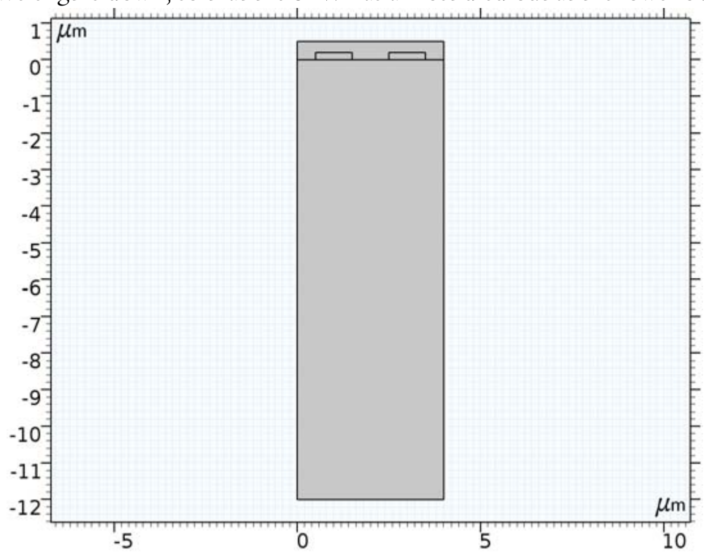


Figure 2: The geometry of the SAW unit cell used in this model. A 500 nm PIB film covers two 1 µm-wide electrodes on top of the $LiNbO_3$ substrate. The substrate domain has a total height of 12 µm.

Set up the model using the predefined Piezoelectric Devices multiphysics interface. In 2D, a Plane Strain assumption is used for the Solid Mechanics interface. Hence the out-of-plane strain component is zero. This should be a valid assumption, considering that the SAW is generated in the plane of the model and hence any variation in the out-of-plane direction can be considered minimal.

BOUNDARY CONDITIONS

In order to define the model, you need to apply structural and electrical boundary conditions.

As one can assume that the surface wave dies off within two to three wavelengths from the surface, the lower boundary is fixed. This enforces a zero structural displacement but does not contribute to any significant reflection from the lower boundary back into the bulk of the substrate as long as we are observing surface waves and in particular Rayleigh waves.

The electrodes have a much higher electrical conductivity compared to PIB and $LiNbO_3$. Hence, one can expect each of the electrodes to be isopotential. This is why you do not need to model the domains that constitute the electrodes but can simply use appropriate boundary conditions on all the outer boundaries of each electrode to indicate what type of isopotential state it is in. The boundaries of the left terminal is set to electrical ground, and those of the right terminal are assigned to a Floating Potential with zero surface charge accumulation. This combination of electrical boundary conditions corresponds to an open circuit configuration, which is typically suitable for sensing applications.

Use periodic boundary conditions to dictate that the electric potential and displacements are the same along both vertical boundaries of the geometry. When using periodic boundary conditions, one needs to ensure that the mesh on the vertical boundaries on the left of the unit cell and the vertical boundaries on the right of the unit cell are identical. This is achieved by first creating the mesh on the vertical boundaries on the left and then using the Copy Edge feature to create the exact same mesh on the vertical boundaries on the right.

All other boundaries are left to the default boundary conditions which are Free for the Solid Mechanics interface and Zero Charge for the Electrostatics interface, respectively.

MATERIAL PROPERTIES

The substrate used in the simulation is YZ-cut $LiNbO_3$ with the following properties (cited in Ref. 2):

- *Elasticity matrix:*

$$\begin{bmatrix} 2.424e11 & 0.752e11 & 0.752e11 & 0 & 0 & 0 \\ & 2.03e11 & 0.573e11 & 0 & 0.085e11 & 0 \\ & & 2.03e11 & 0 & -0.085e11 & 0 \\ & & & 0.752e11 & 0 & 0.085e11 \\ & & & & 0.595e11 & 0 \\ & & & & & 0.595e11 \end{bmatrix}$$

- *Coupling matrix:*

$$\begin{bmatrix} 1.33 & 0.23 & 0.23 & 0 & 0 & 0 \\ 0 & 0 & 0 & -2.5 & 0 & 3.7 \\ 0 & -2.5 & 2.5 & 0 & 3.7 & 0 \end{bmatrix}$$

- *Relative permittivity:*

$$\begin{bmatrix} 28.7 & 0 & 0 \\ & 85.2 & 0 \\ & & 85.2 \end{bmatrix}$$

The density of the PIB film is from Ref. 1. The Poisson's ratio is considered to be 0.48, and the Young's modulus is set to 10 GPa.

The adsorption of DCM gas is represented as a slight increase of the overall density of the PIB film as shown in the following expression.

$$\rho = \rho_{PIB} + switch \cdot \rho_{\mathrm{DCM,PIB}}$$

In this model, you use a parameter, *switch*, whose value can be either 0 or 1. This allows to solve the model for two cases; once without the effect of adsorbed gas and once with the effect of the adsorbed DCM gas in PIB.

When the sensor is exposed to 100 ppm of DCM in air at atmospheric pressure and room temperature, the "partial density" of DCM in the PIB film can be calculated from

$$\rho_{\mathrm{DCM,PIB}} = KMc$$

where $K = 10^{1.4821}$ (Ref. 1) is the air/PIB partition coefficient for DCM, M is its molar mass, and c

$$c = (c_0 p)/(RT)$$

is its concentration in air. The DCM concentration, c in moles/m^3 is computed using the Gas Law. Here c_0 is the concentration in parts per million, p is the pressure, T is the temperature, and R is the gas constant. Any effects of the DCM adsorption on the material properties other than the density are neglected.

Most of the material properties and factors affecting them have been parametrized as shown in Table 1. This easily allows the model to be adapted for other materials and operating conditions.

TABLE 1: LIST OF PARAMETERS

NAME	EXPRESSION	DESCRIPTION
p	1[atm]	Air pressure
T	25[degC]	Air temperature
c0	100	DCM concentration in ppm
c_DCM_air	1e-6*c0*p/(R_const*T)	DCM concentration in air
M_DCM	84.93[g/mol]	Molar mass of DCM
K	10^1.4821	PIB/air partition constant for DCM
rho_DCM_PIB	K*M_DCM*c_DCM_air	Mass concentration of DCM in PIB
rho_PIB	0.918[g/cm^3]	Density of PIB
E_PIB	10[GPa]	Young's modulus of PIB
nu_PIB	0.48	Poisson’s ratio of PIB
eps_PIB	2.2	Relative permittivity of PIB
switch	0	Switch for adding DCM density
vR	3488[m/s]	Rayleigh wave velocity
width	4[um]	Width of unit cell
f0	vR/width	Estimated SAW frequency
t_PIB	0.5[um]	PIB thickness

ESTIMATING THE SAW FREQUENCY

The use of periodic boundary condition implies that the frequencies of interest correspond to wavelengths that are integer fractions of the width of the geometry. The lowest SAW eigenmode has its wavelength equal to the width of the geometry, that is, 4 μm. Using this along with the Rayleigh wave velocity for the given piezoelectric substrate material, one can find an estimate of the resonance frequency of interest. The information can be used in the eigenfrequency solver, which helps it to find out the resonance frequencies close to this estimated number. In this model, you use a YZ-cut $LiNbO_3$ whose Rayleigh wave velocity (vR) is around 3488 m/s. This gives an estimate of the lowest SAW frequency (f0) to be 872 MHz.

Results and Discussion

The anti-resonance and resonance frequencies evaluate to approximately 850 MHz and 855 MHz, respectively. Figure 3 and Figure 4 show the corresponding SAW modes. Figure 5 and Figure 6 show the electric potential distribution characteristics for these solutions.

Exposing the sensor to a 100 ppm concentration of DCM in air leads to a resonance frequency shift of approximately 200 Hz downwards. This is computed by evaluating the resonance frequency before and after adding the density of adsorbed DCM to that of the PIB domain.

Note that the computational mesh is identical in both these solutions. This implies that the relative error of the frequency shift is similar to that of the resonance frequency itself. Thus, the shift is accurately evaluated despite being a few magnitudes smaller than the absolute error of the resonance frequency.

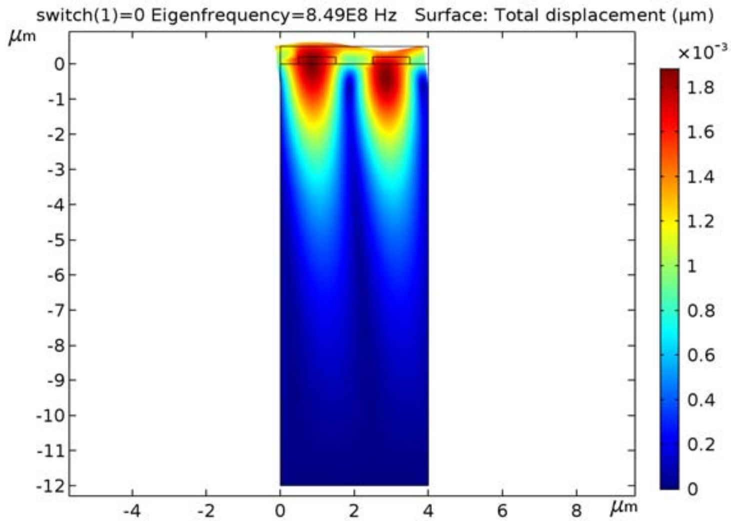


Figure 3: Deformed shape plot of the anti-resonance SAW mode.

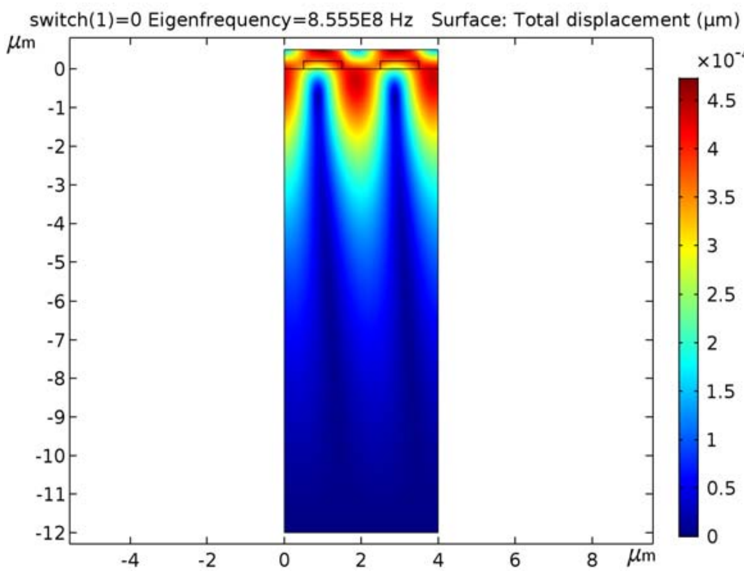


Figure 4: Deformed shape plot of the resonance SAW mode.

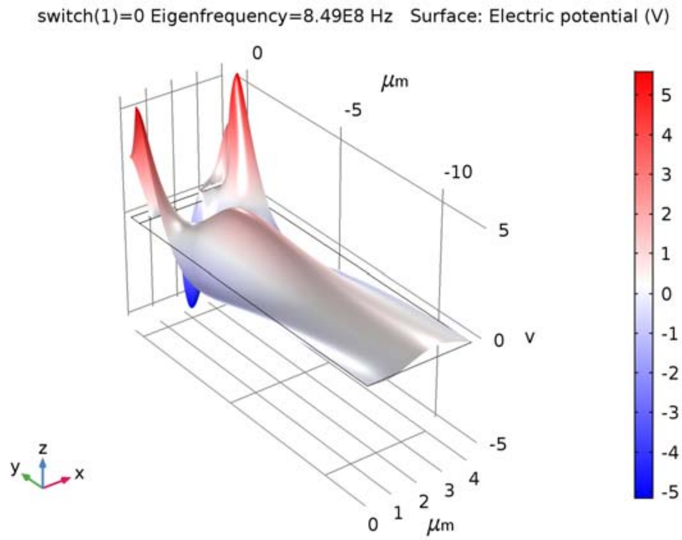


Figure 5: Electric potential distribution and deformations at anti-resonance, anti-symmetric with respect to the center of each electrode.

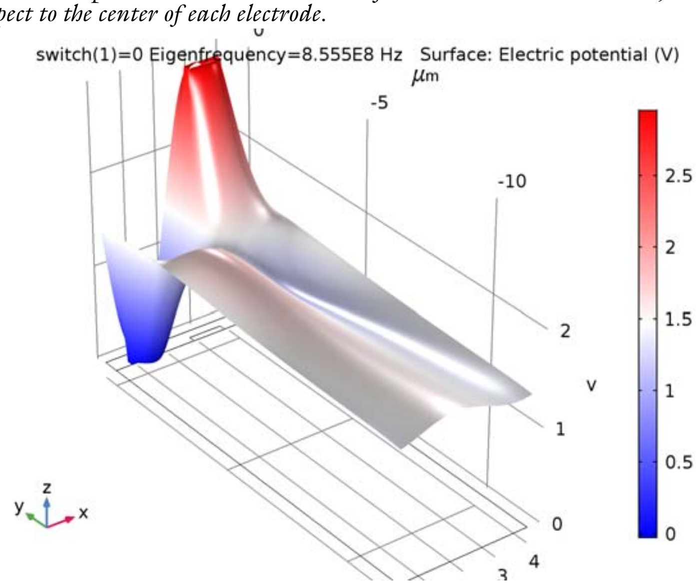


Figure 6: Electric potential distribution at resonance, symmetric with respect to the center of each electrode.

References

1. K. Ho, E. R. Lindgren, K. S. Rawlinson, L. K. McGrath, and J. L. Wright, "Development of a Surface Acoustic Wave Sensor for In-Situ Monitoring of Volatile Organic Compounds," *Sensor*s vol. 3, pp. 236–247, 2003.

2. S. Ahmadi, F. Hassani, C. Korman, M. Rahaman, and M. Zaghloul, "Characterization of Multi- and Single-layer Structure SAW Sensor," *Sensors 2004*, *Proceedings of IEEE*, vol. 3, pp. 1129–1132, 2004.

Application Library path: `MEMS_Module/Sensors/saw_gas_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 **2D**.
2. In the **Select Physics** tree, select **Structural Mechanics>Piezoelectric Devices**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Eigenfrequency**.
6. Click **Done**.

GLOBAL DEFINITIONS

For quicker modeling, load the parameters from a file.

Parameters

1. On the **Home** toolbar, click **Parameters**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.
3. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `saw_gas_sensor_parameters.txt`.

GEOMETRY 1

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**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

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

4 In the **Height** text field, type `3*width+t_PIB`.

5 Locate the **Position** section. In the **y** text field, type `-3*width`.

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

Layer name	Thickness (µm)
Layer 1	t_PIB

7 Select the **Layers on top** check box.

8 Clear the **Layers on bottom** check box.

9 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `width/4`.

4 In the **Height** text field, type `0.4*t_PIB`.

5 Locate the **Position** section. In the **x** text field, type `width/8`.

6 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.

Copy 1 (copy1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **r2** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **x** text field, type `width/2`.

5 Click **Build All Objects**.

Before creating materials, define the domains where each physics apply.

SOLID MECHANICS (SOLID)

Piezoelectric Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Piezoelectric Material 1**.

2 In the **Settings** window for **Piezoelectric Material**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.

ELECTROSTATICS (ES)

On the **Physics** toolbar, click **Solid Mechanics (solid)** and choose **Electrostatics (es)**.

The electrical equations are not solved in the aluminum electrodes because they are assumed to be perfect conductors compared to LiNbO3 and PIB, and hence we consider them to be equipotential regions.

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

2 Select Domains 1 and 2 only.

Charge Conservation, Piezoelectric 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electrostatics (es)** click **Charge Conservation, Piezoelectric 1**.

2 Select Domain 1 only.

MATERIALS

Material 1 (mat1)

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

2 In the **Settings** window for **Material**, type `LiNbO3` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

4 Select Domain 1 only.

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

Property	**Name**	**Value**	**Unit**	**Property group**
Elasticity matrix (Ordering: xx, yy, zz, yz, xz, xy)	cE	{242.4[GPa], 75.2[GPa],203[GPa], 75.2[GPa],57.3[GPa], 203[GPa],0,0,0, 75.2[GPa],0,8.5[GPa], -8.5[GPa],0, 59.5[GPa],0,0,0, 8.5[GPa],0,59.5[GPa]}	Pa	Stress-charge form
Coupling matrix	eES	{1.33,0,0,0.23,0, -2.5,0.23,0,2.5,0, -2.5,0,0,0,3.7,0,3.7, 0}	C/m²	Stress-charge form
Relative permittivity	epsilonrS	{28.7,85.2,85.2}	1	Stress-charge form
Density	rho	4647	kg/m³	Basic

Alternately, you can click the **Edit** button below the **Output properties** table and use the matrix inputs to enter cE, e, and epsilonrS according to the material data shown under the **Model Definition** section.

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

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

3 Select Domain 2 only.

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

Property	**Name**	**Value**	**Unit**	**Property group**
Young's modulus	E	E_PIB	Pa	Basic
Poisson's ratio	nu	nu_PIB	1	Basic
Density	rho	rho_PIB+switch* rho_DCM_PIB	kg/m³	Basic
Relative permittivity	epsilonr	eps_PIB	1	Basic

ADD MATERIAL

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

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

3 In the tree, select **Built-In>Aluminum**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Aluminum (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat3)**.

2 Select Domains 3 and 4 only.

3 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

On the **Physics** toolbar, click **Electrostatics (es)** and choose **Solid Mechanics (solid)**.

Fixed Constraint 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

2 Select Boundary 2 only.

Periodic Condition 1

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Connections>Periodic Condition**.

2 Select Boundaries 1, 3, 16, and 17 only.

This defines periodic condition for the **Solid Mechanics** physics. By default, the periodicity type is set to **Continuity** for all dependent variables.

ELECTROSTATICS (ES)

Ground the left electrode by applying boundary condition on its edges.

Ground 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electrostatics (es)** and choose **Ground**.

2 Select Boundaries 6–9 only.

Apply a **Floating Potential** on the edges of the right electrode.

Floating Potential 1

1 In the **Model Builder** window, right-click **Electrostatics (es)** and choose **Floating Potential**.

2 Select Boundaries 11–14 only.

Define periodic boundary condition for the **Electrostatics** physics.

Periodic Condition 1

1 Right-click **Electrostatics (es)** and choose **Periodic Condition**.

2 Select Boundaries 1, 3, 16, and 17 only.

MESH 1

Edge 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Edge**.

2 Select Boundaries 1 and 3 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Edge 1** and choose **Distribution**.

2 Select Boundary 3 only.

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

4 In the **Number of elements** text field, type 4.

Edge 1

Define an increasing mesh size along the thickness of the piezoelectric substrate.

Distribution 2

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

2 Select Boundary 1 only.

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

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type 25.

6 In the **Element ratio** text field, type 25.

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

8 Click **Build Selected**.

Copy Edge 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Copy Edge**.

2 Select Boundaries 1 and 3 only.

3 In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.

4 Select the **Active** toggle button.

5 Select Boundaries 16 and 17 only.

Free Quad 1

1 Right-click **Mesh 1** and choose **Free Quad**.

2 In the **Settings** window for **Free Quad**, locate the **Domain Selection** section.

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

4 Select Domains 2–4 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Quad 1** and choose **Size**.

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

3 Click the **Custom** button.

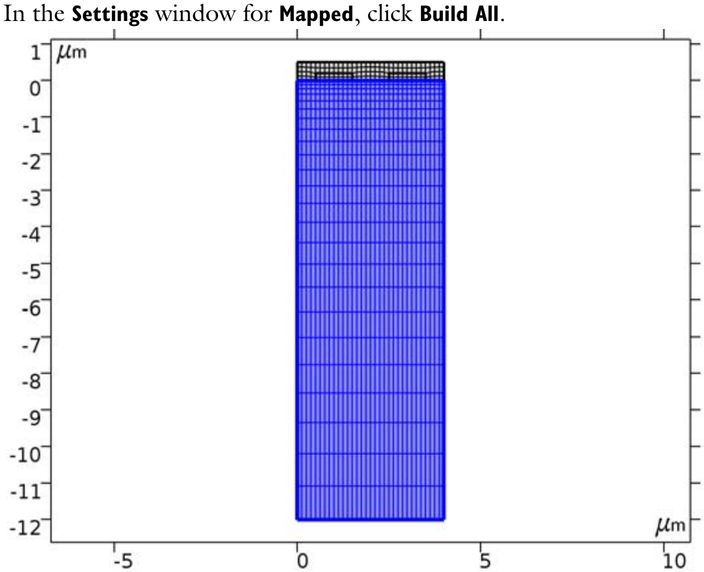
4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

5 In the associated text field, type `t_PIB/4`.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, click **Build All**.



STUDY 1

Set up a **Parametric Sweep** to solve the model with and without the adsorbed species on the sensor. The parameter switch is used to solve the model once without adding the density of DCM and once with the added density of adsorbed DCM in PIB.

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
switch	0 1	

Search for eigenfrequencies near the estimated resonance frequency.

Step 1: Eigenfrequency

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

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

3 Select the **Search for eigenfrequencies around** check box.

4 In the associated text field, type `f0`.

5 On the **Study** toolbar, click **Compute**.

RESULTS

Mode Shape (solid)

The default plot shows the anti-resonance SAW mode near 850 MHz. Adjust the deformed shape plot to see the effect of the wave localization near the surface.

1 In the **Model Builder** window, under **Results** click **Mode Shape (solid)**.

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

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

4 In the **Model Builder** window, expand the **Mode Shape (solid)** node.

Deformation

1 In the **Model Builder** window, expand the **Results>Mode Shape (solid)>Surface 1** node, then click **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box.

4 In the associated text field, type `100`.

5 On the **Mode Shape (solid)** toolbar, click **Plot**.

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

Compare the resulting plot to that in Figure 3.

Mode Shape (solid)

Plot the second mode shape near 855 MHz, which is the resonance SAW mode.

1 In the **Model Builder** window, under **Results** click **Mode Shape (solid)**.

2 Click **Plot**.

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

4 From the **Eigenfrequency (Hz)** list, choose **8.555E8**.

5 On the **Mode Shape (solid)** toolbar, click **Plot**.

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

Compare the resulting plot to that in Figure 4.

Electric Potential (es)

To visualize the electric potential distribution for the eigenmodes, follow these steps.

Surface 1

1 In the **Model Builder** window, expand the **Results>Electric Potential (es)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **WaveLight**.

Electric Potential (es)

1 Right-click **Results>Electric Potential (es)>Surface 1** and choose **Height Expression**.

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

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

4 On the **Electric Potential (es)** toolbar, click **Plot**.

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

6 From the **Eigenfrequency (Hz)** list, choose **8.555E8**.

7 On the **Electric Potential (es)** toolbar, click **Plot**.

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

Derived Values

To see all computed eigenfrequencies as a table, follow these steps.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

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

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

4 From the **Table columns** list, choose **Outer solutions**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Solver>freq - Frequency**.

6 Click **New Table**.

TABLE

1 Go to the **Table** window.

2 Click **Full Precision** in the window toolbar.

The first 6 digits of the eigenfrequency are the same. Subtracting the new value from the previous value shows that the eigenfrequency with gas exposure is lower by 200 Hz.

Piezoelectric Shear-Actuated Beam

Introduction

This example performs a static analysis on a piezoelectric actuator based on the movement of a cantilever beam, using the Piezoelectric Devices predefined multiphysics interface. Inspired by work done by V. Piefort (Ref. 1) and A. Benjeddou (Ref. 2), it models a sandwich beam using the shear mode of the piezoelectric material to deflect the tip.

Model Definition

GEOMETRY

The model consists of a 100-mm long sandwiched cantilever beam (Figure 1).

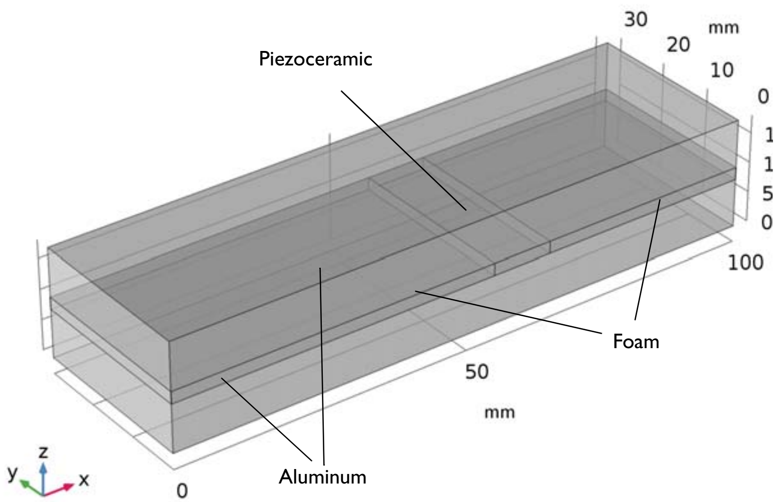


Figure 1: The shear bender geometry. Note that a piezoceramic material replaces part of the foam core.

This beam is composed of a 2-mm thick flexible foam core sandwiched by two 8-mm thick aluminum layers. Further, the device replaces part of the foam core with a 10-mm long piezoceramic actuator that is positioned between x = 55 mm and x = 65 mm. The cantilever beam is orientated along the global x-axis.

BOUNDARY CONDITIONS

- *Solid Mechanics:* the cantilever beam is fixed at its surfaces at $x = 0$; all other surfaces are free.
- *Electrostatics:* The system applies a 20 V potential difference between the top and bottom surfaces of the piezoceramic domain (Figure 2). This gives rise to an electric field perpendicular to the poling direction (x direction) and thus induces a transverse shear strain.

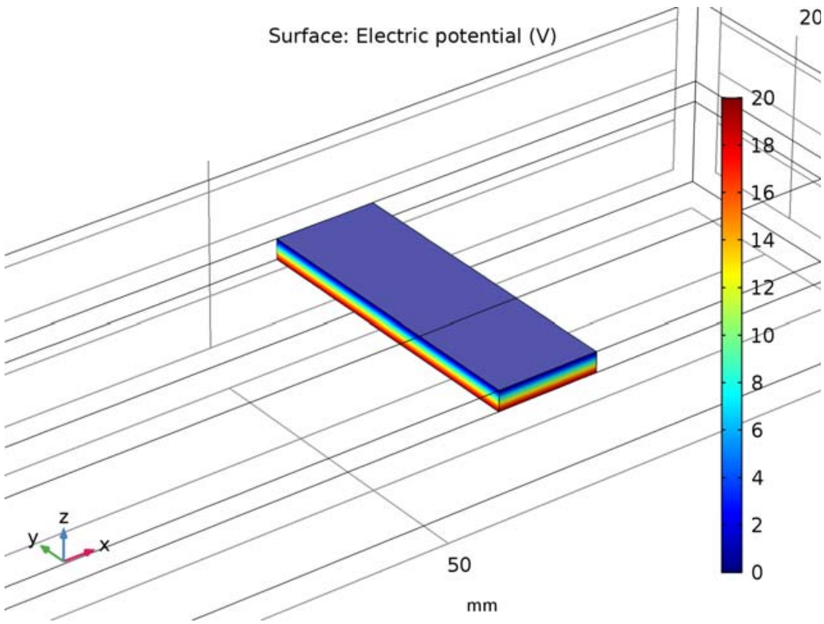


Figure 2: Applied voltage through the piezoelectric material

MATERIAL PROPERTIES

The following table lists the material properties for the aluminum layers and the foam core:

PROPERTY	ALUMINUM	FOAM	PIEZOCERAMIC
E	70 GPa	35.3 MPa	-
ν	0.35	0.383	-
ρ	2700 kg/m^3	32 kg/m^3	7500 kg/m^3

Aluminum is available as a predefined material, whereas you must define the foam material manually.

The piezoceramic material in the actuator, PZT-5H, is already defined in the material library. Thus, you do not need to enter the components of the elasticity matrix, c_E, the piezoelectric coupling matrix, e, or the relative permittivity matrix, ε_{rS}.

Results

The shear deformation of the piezoceramic core layer and the flexible foam layer induce a bending action. Figure 3 shows the resulting tip deflection. The model calculates this deflection as 83 nm, a result that agrees well with those of Ref. 1 and Ref. 2.

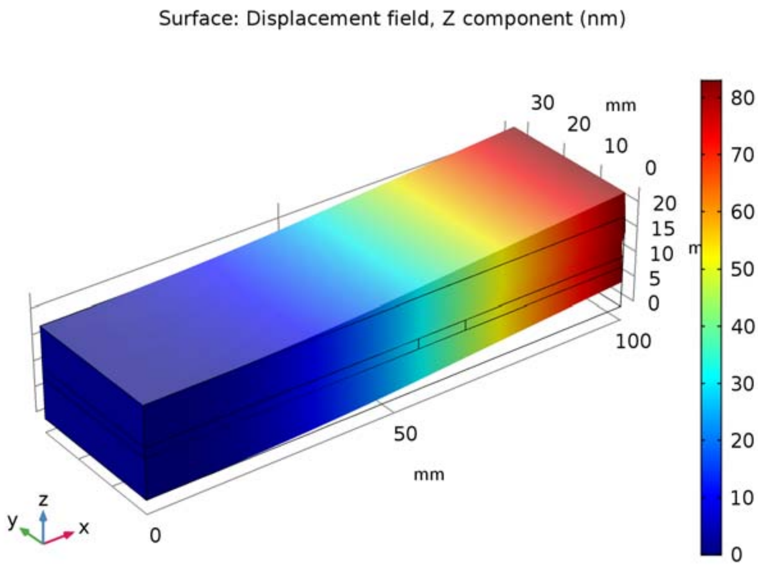


Figure 3: Tip deflection with the piezoceramic positioned at x = 60 mm.

Notes About the COMSOL Implementation

The matrix components for the piezoelectric material properties refer to a coordinate system, where the poling direction is the z direction. Because the poling direction of the piezoceramic actuator in this model is aligned with the x-axis, you need to use a local coordinate system in the material settings to rotate the piezoceramic material.

More specifically, you define a local coordinate system that is rotated 90 degrees about the global y-axis. Then, you use this coordinate system in the piezoelectric material settings to rotate the material so that the polarization direction is aligned with the x-axis (Figure 4).

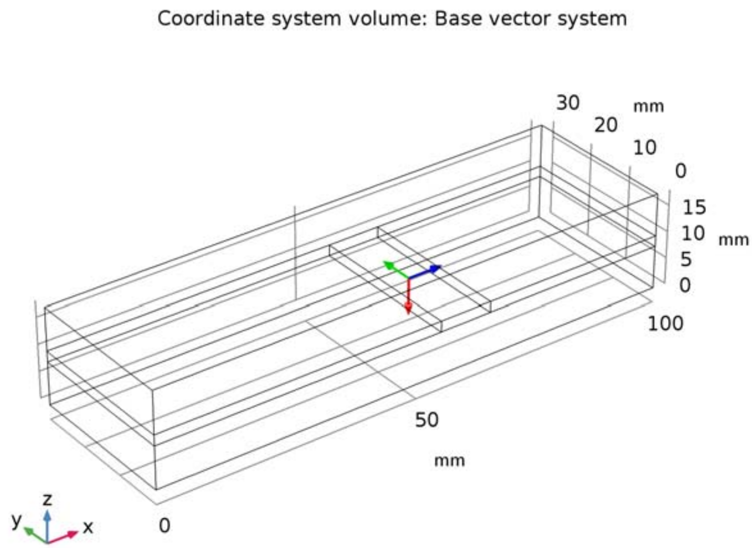


Figure 4: Definition of local coordinate system to define the piezoelectric orientation. The material is poled along the local x3 direction (blue arrow).

References

1. V. Piefort, *Finite Element Modelling of Piezoelectric Active Structures*, Ph.D. thesis, Université Libre de Bruxelles, Belgium, Dept. Mechanical Engineering and Robotics, 2001.

2. A. Benjeddou, M.A. Trindade, and R. Ohayon, *A Unified Beam Finite Element Model for Extension and Shear Piezoelectric Actuation Mechanisms*, CNAM (Paris, France), Structural Mechanics and Coupled Systems Laboratory, 1997.

Application Library path: `MEMS_Module/Piezoelectric_Devices/shear_bender`

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>Piezoelectric Devices**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

6 Click **Done**.

GEOMETRY 1

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 **mm**.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

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

3 In the **Width** text field, type 100.

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

5 In the **Height** text field, type 18.

6 Right-click **Block 1 (blk1)** and choose **Build Selected**.

Block 2 (blk2)

1 On the **Geometry** toolbar, click **Block**.

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

3 In the **Width** text field, type 100.

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

5 In the **Height** text field, type 2.

6 Locate the **Position** section. In the **z** text field, type 8.

7 Click to expand the **Layers** section. Find the **Layer position** subsection. Select the **Left** check box.

8 Clear the **Bottom** check box.

9 In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	55
Layer 2	10

10 Click **Build All Objects**.

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

The model geometry is now complete.

12 Click the **Transparency** button on the **Graphics** toolbar.

The geometry in the Graphics window should now look like that in Figure 1.

13 Click the **Transparency** button on the **Graphics** toolbar.

DEFINITIONS

Define a coordinate system whose third axis is aligned with the global *x*-axis, that is, the polarization direction of the piezoceramic material. Choose the second axis to be parallel to the global y-axis.

Base Vector System 2 (sys2)

1 On the **Definitions** toolbar, click **Coordinate Systems** and choose **Base Vector System**.

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

3 Find the **Base vectors** subsection. In the table, enter the following settings:

	x	y	z
x1	0	0	-1
x3	1	0	0

Leave the other components at their default values. You will use this coordinate system in the piezoelectric material settings.

4 Find the **Simplifications** subsection. Select the **Assume orthonormal** check box.

ELECTROSTATICS (ES)

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

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

3 Click **Clear Selection**.

4 Select Domain 4 only.

SOLID MECHANICS (SOLID)

On the **Physics** toolbar, click **Electrostatics (es)** and choose **Solid Mechanics (solid)**.

Piezoelectric Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Piezoelectric Material 1**.

2 In the **Settings** window for **Piezoelectric Material**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 4 only.

5 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Base Vector System 2 (sys2)**.

MATERIALS

For the aluminum layers, use a library material.

ADD MATERIAL

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

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

3 In the tree, select **MEMS>Metals>Al - Aluminum / Aluminium**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Al - Aluminum / Aluminium (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Al - Aluminum / Aluminium (mat1)**.

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

3 Click **Clear Selection**.

4 Select Domains 1 and 3 only.

For the foam core, specify the material properties by hand.

Material 2 (mat2)

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

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

3 Select Domains 2 and 5 only.

Foam (mat2)

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

Property	**Name**	**Value**	**Unit**	**Property group**
Young's modulus	E	`35.3[MPa]`	Pa	Basic
Poisson's ratio	nu	`0.383`	1	Basic
Density	rho	32	kg/m^3	Basic

ADD MATERIAL

1 Go to the **Add Material** window.

The piezoceramic PZT-5H is available as a predefined material.

2 In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-5H)**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Lead Zirconate Titanate (PZT-5H) (mat3)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials>Foam (mat2)** node, then click **Component 1 (comp1)>Materials>Lead Zirconate Titanate (PZT-5H) (mat3)**.

2 Select Domain 4 only.

3 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

2 Select Boundaries 1, 4, and 7 only.

ELECTROSTATICS (ES)

Electric Potential 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electrostatics (es)** and choose **Electric Potential**.

2 Select Boundary 16 only.

3 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

4 In the V_0 text field, type 20.

Ground 1

1 In the **Model Builder** window, right-click **Electrostatics (es)** and choose **Ground**.

2 Select Boundary 17 only.

MESH 1

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Swept**.

2 Right-click **Swept 1** and choose **Distribution**.

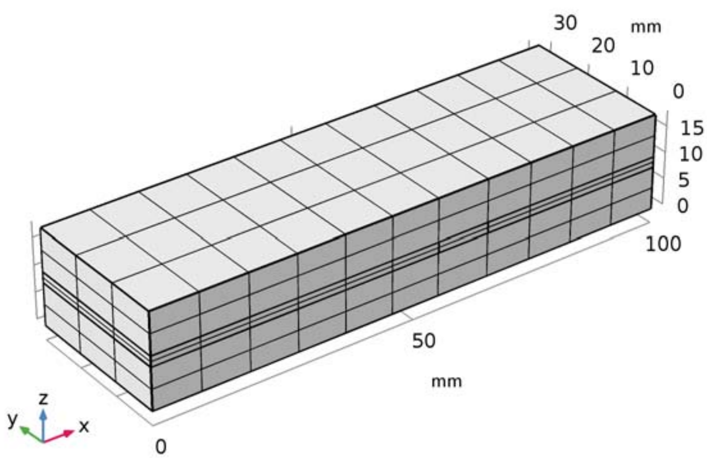
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 2.

5 Click **Build All**.

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

The mesh consists of 198 hexahedral elements.



STUDY 1

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

RESULTS

Stress (solid)

Replace the default stress plot by displacement to reproduce the plot shown in Figure 3.

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement (solid)` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Displacement (solid)** node, then click **Surface 1**.

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>Solid Mechanics> Displacement>Displacement field (material and geometry frames)>w - Displacement field, Z component**.

3 Locate the **Expression** section. From the **Unit** list, choose **nm**.

4 On the **Displacement (solid)** toolbar, click **Plot**.

5 Click **Go to Default View**.

Multislice 1

In the **Model Builder** window, expand the **Electric Potential (es)** node.

Surface 1

1 Right-click **Multislice 1** and choose **Delete**.

2 In the **Model Builder** window, under **Results** right-click **Electric Potential (es)** and choose **Surface**.

3 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electrostatics>Electric>V - Electric potential**.

4 On the **Electric Potential (es)** toolbar, click **Plot**.

Zoom in to find a plot similar to Figure 2.

5 Click the **Zoom In** button on the **Graphics** toolbar.

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

Show the base vector that defines the polarization of the piezoelectric material, shown on Figure 4.

3D Plot Group 3

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

2 In the **Settings** window for **3D Plot Group**, type `PZT coordinate system` in the **Label** text field.

Coordinate System Volume 1

1 On the **PZT coordinate system** toolbar, click **More Plots** and choose **Coordinate System Volume**.

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

3 From the **Coordinate system** list, choose **Base Vector System 2 (sys2)**.

4 Locate the **Positioning** section. Find the **x grid points** subsection. From the **Entry method** list, choose **Coordinates**.

5 In the **Coordinates** text field, type `60`.

6 Find the **y grid points** subsection. In the **Points** text field, type `1`.

7 Find the **z grid points** subsection. In the **Points** text field, type `1`.

8 On the **PZT coordinate system** toolbar, click **Plot**.

Squeeze-Film Gas Damping of a Vibrating Disc

Introduction

This benchmark model computes the damping force acting on a vibrating disc. The disc is in close proximity to a stationary surface and the damping results from the squeezing of a thin film of gas between the two surfaces. The squeezing action forces out the gas from between the two plates resulting in a damping force that acts to prevent mechanical contact between the two surfaces. The opposite effect takes place when the surfaces move away from each other as gas is drawn back into the bearing.

This model examines the effect of the periodic motion of the disc on the flow developed, including the pressure in the gas and the resulting damping forces. Small amplitude motion is analyzed using a linear frequency domain simulation. A nonlinear transient analysis is performed for small to large amplitude motion. The calculated film pressure and load carrying capacity are compared with analytical results.

Model Definition

The Thin-Film Flow, Edge interface is used to model the gas film on a flat circular plate. The model is 1D axisymmetric since the film pressure only varies radially. When Thin-Film Flow is assigned to a boundary, this boundary represents a reference surface in the physical device. In practice a small gap exists at the boundary and two impermeable structures, the wall and the base, are located either side of it. Figure 1 shows the configuration of the base and the wall in an arbitrary problem, and defines a number of terms used in the interface.

In this example, the model geometry is 1D axisymmetric and consists of a single line, with length set to the radius of the circular disc. The line is located at the origin and aligned with the r-axis. The base is coincident with the reference surface. A pressure is generated in the bearing by a periodic velocity of the wall in a direction normal to the wall.

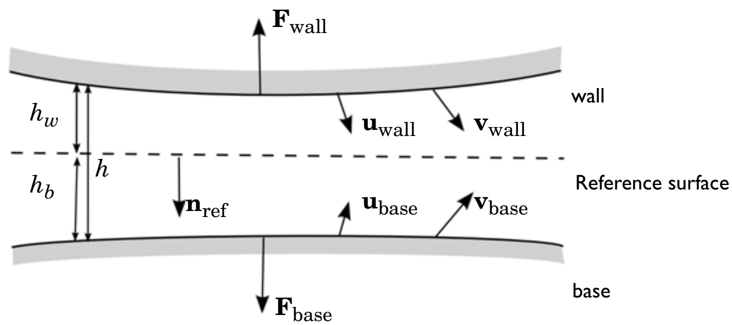


Figure 1: An example illustrating a typical configuration for thin-film flow.

For non-slip boundary conditions at the wall and the base, the modified Reynolds equation takes the following form for a general frequency domain problem:

$$\begin{aligned} p_{tot}(\mathbf{v}_b \cdot \mathbf{n}_{ref} - \mathbf{v}_w \cdot \mathbf{n}_{ref}) + i\omega h_0 p_f + \nabla_t \cdot (h_0 p_{tot} \mathbf{v}_{av}) \\ -p_{tot}(\mathbf{v}_w \cdot \nabla_t h_w + \mathbf{v}_b \cdot \nabla_t h_b) = 0 \end{aligned} \tag{1}$$

$$\mathbf{v}_{av} = \frac{1}{2}(\mathbf{I} - \mathbf{n}_r \mathbf{n}_r^{\ T})(\mathbf{v}_w + \mathbf{v}_b) - \frac{h_0^{\ 2}}{12\mu} \nabla_t p_f$$

where ρ is the fluid density, μ is its viscosity, h_0 is the mean film height, p_f is the pressure developed as a result of the flow (this is the dependent variable in COMSOL) and p_{tot} is the total pressure (p_{tot}=p_A+p_f, where p_A is the ambient pressure). Other terms are defined in Figure 1. A reference surface with normal $\mathbf{n}_{\text{ref}}$ sits in a narrow gap between a wall and base. In COMSOL the vector $\mathbf{n}_{\text{ref}}$ points into the base and out of the wall. The wall moves with a displacement from its initial position with displacement $\mathbf{u}_{\text{wall}}$ and velocity $\mathbf{v}_{\text{wall}}$. Similarly the base moves from its initial position with displacement $\mathbf{u}_{\text{base}}$ and velocity $\mathbf{v}_{\text{base}}$. The compression of the film results in an excess pressure, p_f, above the ambient pressure, p_A, and a gas velocity in the gap. At a point on the reference surface the average value of the film velocity along a line perpendicular to the surface is given by the in plane vector $\mathbf{v}_{\text{ave}}$. The motion of the gas results in forces on the wall ($\mathbf{F}_{\text{wall}}$) and the base ($\mathbf{F}_{\text{base}}$). The height of the wall above the reference surface is h_w whilst the base is a distance h_b below the reference surface. The total size of the gap is h=h_w+h_b. At a given point in time h_w=h_{w1}−$\mathbf{n}_{\text{ref}} \cdot \mathbf{n}_{\text{wall}}$ and h_b=h_{b1}−$\mathbf{n}_{\text{ref}} \cdot \mathbf{n}_{\text{wall}}$ where h_{w1} and h_{b1} are the initial heights of the wall and base, respectively.

Note that the frequency formulation assumes a small amplitude first order harmonic variation of film pressure, film height and wall velocity at the frequency of interest. The boundary conditions for this model are vanishing pressure due to the flow ($p_f = 0$) at $r = r_0$, where r_0 is the disc radius and symmetry/zero pressure gradient ($dp_f/dr = 0$) at $r = 0$.

For the case of a 1D axisymmetric problem Equation 1 can be greatly simplified to derive a simple closed form analytical solution. Note that these simplifications are not made in the simulation itself and consequently there are slight deviations from the analytic results in the model—these cannot be seen in the plots shown here and are not significant (strictly speaking the simulation is more accurate than the analytic results since no assumptions are made). The motion of the disc is in the vertical direction only, and the gap size is uniform across the disc, so a number of the terms in Equation 1 are zero. In this example, the term $i\omega h p_f$ is quite small compared to other terms and can be neglected for the purpose of deriving an analytical solution. The ambient pressure is 1 atmosphere and correspondingly $p_f \ll p_A$. so $p_{tot} \approx p_A$. Making these assumptions the modified Reynold's equation reduces to:

$$\frac{1}{r}\frac{d}{dr}\left(\frac{r h_0^3}{12\mu}\frac{dp_f}{dr}\right) = v_w \tag{2}$$

Where v_w is the velocity of the wall in the z-direction. With the boundary conditions $p_f = 0$ at $r = r_0$, and $dp_f/dr = 0$ at $r = 0$, Equation 2 can be solved for p_f and is given by (see Ref. 1 for complete derivation):

$$p_{f,an} = -\frac{3\mu v_w}{h_0^3}(r_0^2 - r^2) \tag{3}$$

The total analytical vertical load on the disc is then given by (see Ref. 1 for complete derivation):

$$F_{an} = \int_0^{r_0} 2\pi r p_f dr = -\frac{3\pi\mu r_0^4 v_w}{2h_0^3}$$

A frequency domain analysis is appropriate for small amplitude periodic motion of the bearing wall. For large amplitude periodic motion (where the amplitude of the motion becomes comparable to the gap size), a transient analysis is necessary, to capture the nonlinearities in the model. Again approximations are required to derive an analytic result for the transient simulation (but are not necessary for the model).

For a transient model the modified Reynolds equation takes the form:

$$\frac{\partial}{\partial t}(p_{tot}h) + \nabla_t \cdot (hp_{tot}\mathbf{v}_{av}) - p_{tot}(\mathbf{v}_w \cdot \nabla_t h_w + \mathbf{v}_b \cdot \nabla_t h_b) = 0 \qquad (4)$$

$$\mathbf{v}_{av} = \frac{1}{2}(\mathbf{I} - \mathbf{n}_r\mathbf{n}_r^{\;T})(\mathbf{v}_w + \mathbf{v}_b) - \frac{h^2}{12\mu}\nabla_t p_f$$

For periodic motion of the wall, the total film height h is $h(t)=h_0+\Delta h\sin(2\pi ft)$, where Δh is the amplitude and f is the frequency of wall periodic motion. The wall velocity v_{w} is then given by $v_{\mathrm{w}}(t) = (2\pi f\Delta h)\cos(2\pi ft)$. Once again making the assumption that $p_{tot} \approx p_A$ and noting that a number of these terms are zero for vertical motion of a parallel disc:

$$\frac{1}{r}\frac{d}{dr}\left(\frac{rh(t)^3}{12\mu}\frac{dp_f}{dr}\right) = \mathrm{v}_w(t)$$

Following the derivation of total analytical vertical load for the frequency domain analysis, the total vertical load on the disc for the transient model is given by

$$F_{an}(t) = \int_0^{r_0} 2\pi r p_f(t)dr = -\frac{3\pi\mu r_0^{\;4}\mathrm{v}_w(t)}{2h(t)^3}$$

The model compares the analytical values of film pressure and total vertical load against the values computed by using the Thin-Film Flow, Edge interface. The results are found to be in agreement with the analytical solutions.

Results and Discussion

The values of radial film pressure in the gas film for the frequency domain analysis are plotted in Figure 2. As expected, film pressure magnitude is maximum at the center of the circular disc and drops off to zero where the gas film exits the bearing geometry. Figure 2 also compares the numerical radial film pressure values with analytical values calculated using Equation 3. The calculated results agree well with the analytical results. Figure 3 shows an arrow plot of the fluid load per unit area on the circular disc. These values correspond to the film pressure since the vertical load is significantly larger than the radial load. Figure 4 shows the variation of film height (gap) with respect to time for different values of the amplitude of harmonic film height. The corresponding variation of film pressure and total vertical load on the circular disc is shown in Figure 5 and Figure 6, respectively. For larger amplitude of harmonic film height the response in terms of both film pressure and total load are nonlinear with respect to the applied harmonic motion of the circular plate. Such nonlinearity is appropriately calculated by performing a transient analysis. Figure 6 also compares analytical and calculated time dependent values of the total load on the circular disc. The plot indicates that the calculated values agree well with the analytical values.

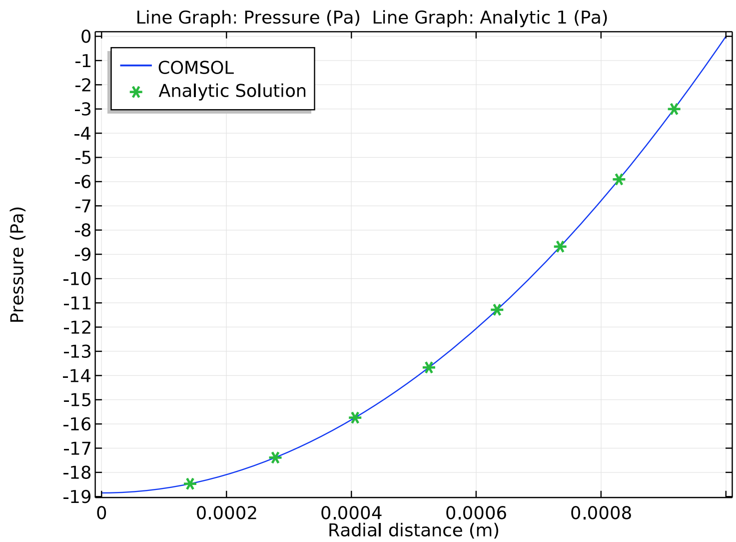


Figure 2: Film pressure vs radial distance from the center of the circular disc. The results computed by COMSOL are shown as the continuous curve and the analytical result is shown with green symbols.

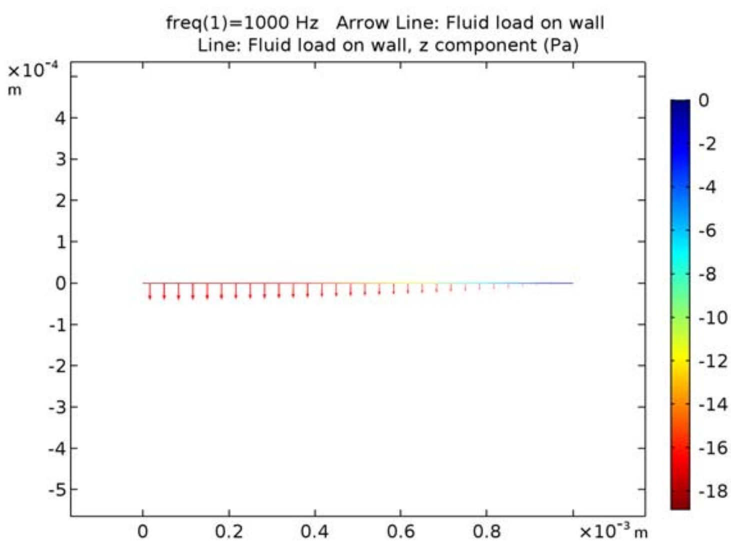


Figure 3: Radial distribution of the film load on the wall

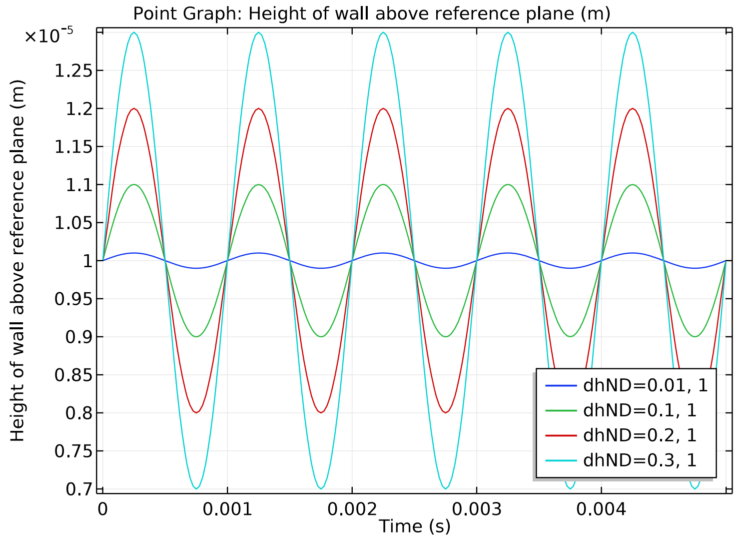


Figure 4: Film height variation vs time for different values of amplitude of harmonic film height.

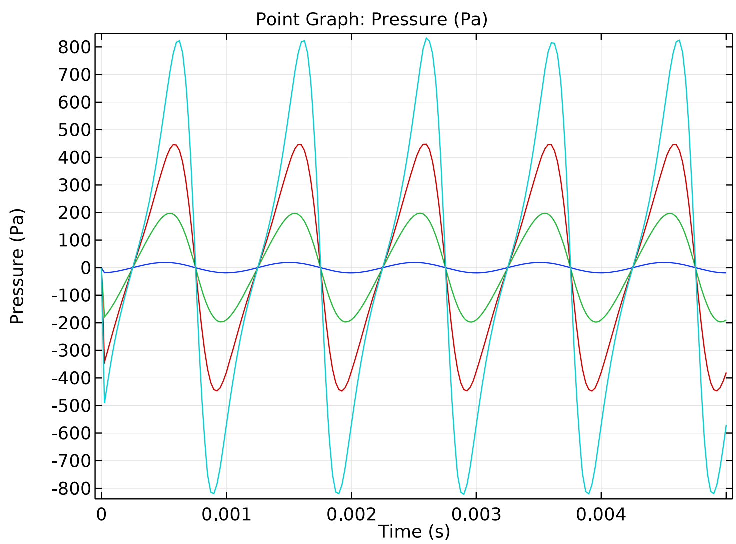


Figure 5: Film pressure vs time for different values of amplitude of harmonic film height. For higher values of film height amplitude the film pressure varies nonlinearly with respect to the film height.

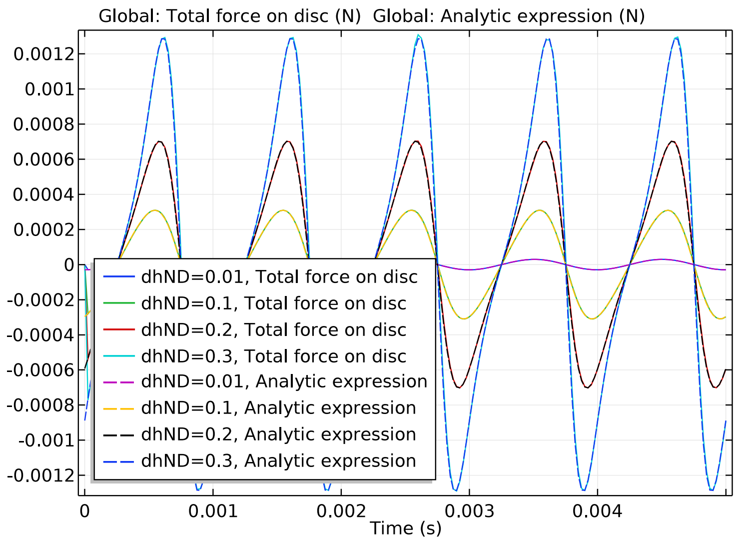


Figure 6: Total vertical load on the circular disc vs time. for different values of amplitude of harmonic film height. The results computed by COMSOL are shown by solid continuous curve and the analytical result is shown by dashed continuous curves. Similar to the film pressure, for higher values of film height amplitude the total load varies nonlinearly with respect to the film height.

Reference

1. B.J. Hamrock, S.R. Schmid, and B.O. Jacobson, *Fundamentals of Fluid Film Lubrication*, Marcel Dekker, 2004.

This model is based on the discussion entitled *Parallel-Surface Bearing of infinite width* in section 12.2 of the above reference.

Application Library path: `MEMS_Module/Sensors/squeeze_film_disc`

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 **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Fluid Flow>Thin-Film Flow>Thin-Film Flow, Edge (tffs)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Frequency Domain**.

6 Click **Done**.

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:

Name	Expression	Value	Description
r0	1[mm]	0.001 m	Disc radius
h0	10[um]	1E-5 m	Gap height
dh	dhND*h0	1E-7 m	Change in gap height
dhND	0.01	0.01	Fractional gap height change.
mu0	1e-5[Pa*s]	1E-5 Pa·s	Gas viscosity
f0	1000[Hz]	1000 Hz	Vibration frequency

GEOMETRY 1

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **r** to r0.

5 Click **Build All Objects**.

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

DEFINITIONS

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 **Boundary**.

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

5 Locate the **Advanced** section. Select the **Compute integral in revolved geometry** check box.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Ftot	intop1(tffs.fwallz)	N	Total force on disc
Ftotan	-3*pi*mu0*vf*r0^4/(2*h0^3)	N	Analytic expression
vf	2*pi*f0*dh	m/s	Disc velocity
Ftotantime	-6*pi^2*mu0*f0*r0^4*dh* cos(2*pi*f0*t)/(2*(h0+dh* sin(2*pi*f0*t))^3)	N	Analytic expression

Analytic 1 (an1)

1 On the **Definitions** toolbar, click **Analytic**.

2 In the **Settings** window for **Analytic**, type `Pan` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `-6*pi*mu0*f0*dh*(r0^2-rf^2)/h0^3`.

4 In the **Arguments** text field, type `rf`.

5 Locate the **Units** section. In the **Arguments** text field, type `m`.

6 In the **Function** text field, type `Pa`.

MATERIALS

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Dynamic viscosity	mu	`mu0`	Pa·s	Basic

The density is not required because the modified Reynolds equation is solved.

THIN-FILM FLOW, EDGE (TFFS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Thin-Film Flow, Edge (tffs)**.

2 In the **Settings** window for **Thin-Film Flow, Edge**, locate the **Equation Type** section.

3 From the **Equation type** list, choose **Modified Reynolds equation**.

Fluid-Film Properties 1

Reverse the direction of the geometry normal. Note that this is necessary to ensure that the wall load acts in the vertical direction. See for more details on the orientation of the wall and base with respect to the geometric and reference surface normals. To view the orientation of the reference normal, show the default solver settings and then compute only to the dependent variables stage in the study sequence. Then plot the reference surface normal.

1 In the **Model Builder** window, under **Component 1 (comp1)>Thin-Film Flow, Edge (tffs)** click **Fluid-Film Properties 1**.

2 In the **Settings** window for **Fluid-Film Properties**, click to expand the **Reference surface properties** section.

3 Locate the **Reference Surface Properties** section. From the **Reference normal orientation** list, choose **Opposite direction to geometry normal**.

4 Locate the **Wall Properties** section. In the h_{w1} text field, type `h0`.

5 From the $\mathbf{u}_w$ list, choose **None**.

6 From the $\mathbf{v}_w$ list, choose **User defined**. Specify the vector as

`0`	r
`vf`	z

Fluid-Film Properties 2

1 Right-click **Component 1 (comp1)>Thin-Film Flow, Edge (tffs)>Fluid-Film Properties 1** and choose **Duplicate**.

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

3 In the **Settings** window for **Fluid-Film Properties**, locate the **Wall Properties** section.

4 From the $\mathbf{u}_w$ list, choose **User defined**. Specify the **associated** vector as

`0`	r
`dh*sin(2*pi*f0*t)`	z

5 From the $\mathbf{v}_w$ list, choose **Calculate from wall displacement**.

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 **Element size** list, choose **Extra fine**.

4 Click **Build All**.

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

STUDY 1

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `1000`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Thin-Film Flow, Edge (tffs)>Fluid-Film Properties 2**.

5 Click **Disable**.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)**.

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

RESULTS

1D Plot Group 2

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 **Plot Settings** section.

3 Select the **x-axis label** check box.

4 In the associated text field, type `Radial distance (m)`.

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

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

7 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1 Right-click **1D Plot Group 2** and choose **Line Graph**.

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

3 In the **Settings** window for **Line Graph**, click to expand the **Legends** section.

4 Select the **Show legends** check box.

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
COMSOL

Line Graph 2

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 2** and choose **Line Graph**.

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

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

4 In the **Expression** text field, type `Pan(r)`.

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

7 Locate the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
Analytic Solution

10 On the **1D Plot Group 2** toolbar, click **Plot**.

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

1D Plot Group 2

1 Right-click **1D Plot Group 2** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Radial Pressure` in the **New label** text field.

3 Click **OK**.

Arrow Line 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Model Builder** window, right-click **2D Plot Group 3** and choose **Arrow Line**.

3 In the **Settings** window for **Arrow Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Thin-Film Flow, Edge> Fluid loads>tffs.fwallr,tffs.fwallz - Fluid load on wall**.

4 Locate the **Coloring and Style** section. In the **Number of arrows** text field, type `30`.

5 Select the **Scale factor** check box.

6 Use the slider to adjust the arrow length.

7 On the **2D Plot Group 3** toolbar, click **Plot**.

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

Line 1

1 In the **Model Builder** window, under **Results** right-click **2D Plot Group 3** and choose **Line**.

2 In the **Settings** window for **Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Thin-Film Flow, Edge> Fluid loads>Fluid load on wall>tffs.fwallz - Fluid load on wall, z component**.

3 Locate the **Coloring and Style** section. Select the **Reverse color table** check box.

4 On the **2D Plot Group 3** toolbar, click **Plot**.

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

2D Plot Group 3

1 Right-click **2D Plot Group 3** and choose **Rename**.

2 In the **Rename 2D Plot Group** dialog box, type `Wall Load` in the **New label** text field.

3 Click **OK**.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`abs(Ftot)`	`N`	

4 Click **Evaluate**.

Global Evaluation 2

1 On 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>Definitions>Variables>Ftotan - Analytic expression**.
3 Click **Table 1 - Global Evaluation 1 (abs(Ftot))**.

ROOT

The agreement between the total force computed by COMSOL and the analytic expression is excellent.

Next add a study to solve the problem in the time domain.

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>Time Dependent**.
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: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.
2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3 Click **Range**.
4 In the **Range** dialog box, type `1/(40*f0)` in the **Step** text field.
5 In the **Stop** text field, type `5/f0`.
6 Click **Replace**.
7 In the **Settings** window for **Time Dependent**, click to expand the **Study extensions** section.
8 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
9 Click **Add**.

10 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dhND	0.01 0.1 0.2 0.3	

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

RESULTS

1D Plot Group 5

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 2/Solution 2 (sol2)**.

Point Graph 1

1. Right-click **1D Plot Group 5** and choose **Point Graph**.
2. Select Point 1 only.
3. In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Thin-Film Flow, Edge>Wall and base properties>tffs.hw - Height of wall above reference plane**.
4. On the **1D Plot Group 5** toolbar, click **Plot**.
5. Click to expand the **Legends** section. Select the **Show legends** check box.

1D Plot Group 5

1. In the **Model Builder** window, under **Results** click **1D Plot Group 5**.
2. In the **Settings** window for **1D Plot Group**, click to expand the **Axis** section.
3. Select the **Manual axis limits** check box.
4. In the **y minimum** text field, type `0`.
5. Locate the **Legend** section. From the **Position** list, choose **Lower right**.
6. On the **1D Plot Group 5** toolbar, click **Plot**.
7. Right-click **Results>1D Plot Group 5** and choose **Rename**.
8. In the **Rename 1D Plot Group** dialog box, type `Wall height vs time` in the **New label** text field.
9. Click **OK**.
10. On the **Wall height vs time** toolbar, click **Plot**.

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

The wall height varies sinusoidally, as expected. Except in the case when `dhND` is 0.01, the height varies by a significant fraction of the gap. This means that the equation system is non-linear.

1D Plot Group 6

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 2/Solution 2 (sol2)**.

Point Graph 1

1 Right-click **1D Plot Group 6** and choose **Point Graph**.

2 Select Point 1 only.

3 On the **1D Plot Group 6** toolbar, click **Plot**.

1D Plot Group 6

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 6** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Pressure vs time` in the **New label** text field.

3 Click **OK**.

4 On the **Pressure vs time** toolbar, click **Plot**.

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

Once the wall height varies by a significant fraction of the gap non-linear effects become important. The pressure does not vary sinusoidally any more and it is no longer possible to model the equation straightforwardly in the frequency domain.

When the height variations are small direct comparison with the frequency domain solution is possible. Do this by comparing the maximum value of the pressure in the time domain with the amplitude of the pressure computed in the frequency domain.

Point Evaluation 1

1 On the **Results** toolbar, click **Point Evaluation**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
abs(pfilm)	Pa	

5 Click **Evaluate**.

Point Evaluation 2

1 On the **Results** toolbar, click **Point Evaluation**.

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

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

4 From the **Parameter selection (dhND)** list, choose **From list**.

5 In the **Parameter values (dhND)** list, select **0.01**.

6 Select Point 1 only.

7 Locate the **Data Series Operation** section. From the **Operation** list, choose **Minimum**.

8 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

In this case the time and frequency domain analyses are in good agreement.

RESULTS

1D Plot Group 7

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

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

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

Global 1

1 Right-click **1D Plot Group 7** and choose **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>Definitions>Variables> Ftot - Total force on disc**.

Global 2

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 7** and choose **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>Definitions>Variables> Ftotantime - Analytic expression**.

3 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

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 **Axis** section.

3 Select the **Manual axis limits** check box.

4 In the **x minimum** text field, type `0.003`.

5 In the **x maximum** text field, type `0.005`.

6 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

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

8 In the **Rename 1D Plot Group** dialog box, type `Total force on disc vs time` in the **New label** text field.

9 Click **OK**.

10 On the **Total force on disc vs time** toolbar, click **Plot**.

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

The agreement between the COMSOL result and the analytic expression is good. The force acting on the disc remains periodic but has higher harmonic components at large displacements relative to the gap size.

Created in COMSOL Multiphysics 5.3

Surface Micromachined Accelerometer

Introduction

This example shows how to model a surface micromachined accelerometer in COMSOL, using the electromechanics interface. The example is based on the case study in Ref. 1. The model also demonstrates the new geometry feature of Linked Subsequences, first introduced in COMSOL 5.0. A collection of geometric building blocks can be stored in a source model file as Subsequences. Thereafter other model files can re-use the same building blocks by linking to the Subsequences in the source model file. Each Subsequence can take arguments to generate a building block with specific dimensions and/or number of features. In this model a surface micromachined accelerometer is created from three building blocks, two of which are used multiple times by calling the corresponding Subsequence with different arguments.

Model Definition

The surface micromachined accelerometer is composed of a released proof mass supported by anchored springs at its two ends, together with sensing and self test electrodes extending to the sides. When the device is subject to an acceleration, the restoring force from the springs gives a displacement of the proof mass in proportion to the acceleration. The displacement causes a change in the capacitance between the fixed and moving electrodes. This change in capacitance can be measured with a number of standard circuits.

For acceleration along the axis of the accelerometer, symmetry allows modeling only half of the geometry for faster computation. The three geometric building blocks are the proof mass with attached electrodes (Figure 1), the folded spring (Figure 2), and the fixed electrode array (Figure 3). These building blocks are implemented as Subsequences that take arguments to specify dimensions, orientation, position, and number of features. For example, the proof mass shown in Figure 1 has 7 sense electrodes at the center and 3 self test electrodes at each end. The actual model on the other hand is built with 21 sense electrodes, by calling the same Subsequence with the corresponding argument 21. As another example, Figure 4 shows an electrode array built from the same Subsequence as in Figure 3, with a different set of arguments, resulting in different number of electrodes, dimensions, and orientation of the anchor pads.

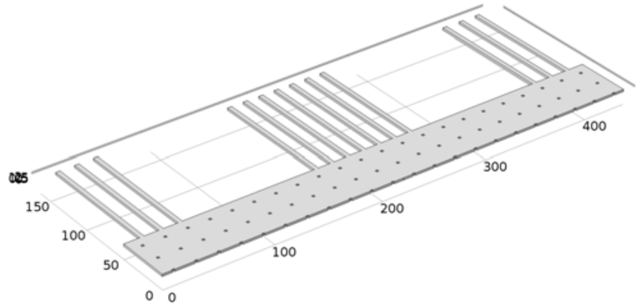


Figure 1: Building block for the proof mass with attached electrodes. Grid scales are in micrometers.

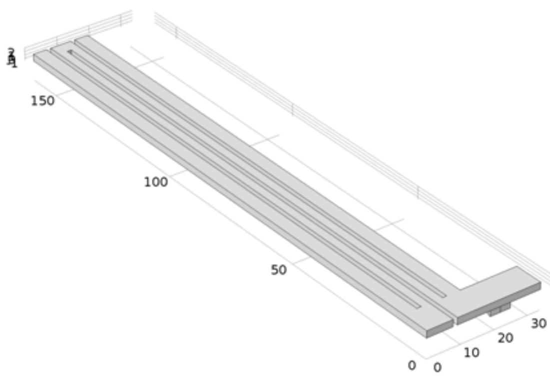


Figure 2: Building block for the folded spring. Grid scales are in micrometers.

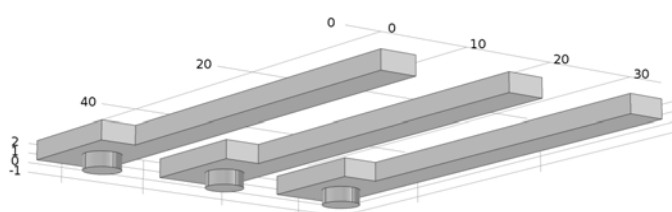


Figure 3: Building block for the fixed electrode array. Grid scales are in micrometers.

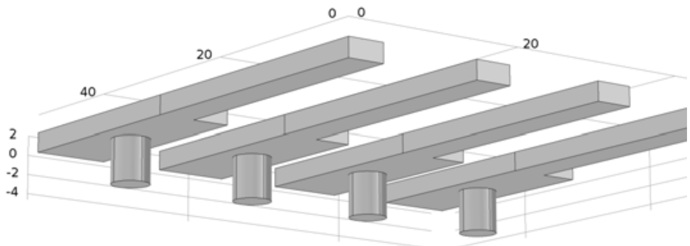


Figure 4: Example of an electrode array built from the same Subsequence as in Figure 3, with a different set of arguments. Grid scales are in micrometers.

The geometry sequence begins by calling the proof mass Subsequence, followed by calling the folded spring Subsequence twice to attach a spring at each end of the proof mass. Subsequently 6 calls to the fixed electrode array Subsequence are made to construct the sense and self test fixed electrodes.

Each Subsequence call contributes to a domain and a boundary selection. This allows easy assignment of domain physics and boundary conditions in the physics interface.

The model uses polysilicon for the building material and includes a rectangular air domain surrounding the polysilicon. The Electromechanics interface models the electric field within the deforming gaps between the electrodes, and applies the appropriate electrostatic forces to the solids, which creates a corresponding structural deformation. The narrowing electrostatics domain results in nonlinear geometrical effects which are included in the Electromechanics interface by default.

The entire polysilicon solid is subject to an acceleration using the Body Load domain physics feature. The (mechanically) fixed electrodes are set at constant potentials, and the proof mass (and its attached electrodes) is at a floating potential whose value will be determined by the position-dependent capacitance (and the applied voltages on the fixed sense electrodes).

Results and Discussion

The first study illustrates the normal operation of the accelerometer by sweeping the applied acceleration from -50 to +50 g and computing the resulting displacement of the proof mass. Figure 5 shows the displacement of the polysilicon domains when the applied acceleration is 50 g. The proof mass (and the attached moving electrodes) moves by about

0.07 micrometer. The anchored spring bases and the fixed electrodes have very little movement. The folded springs have varying displacement along its length as expected.

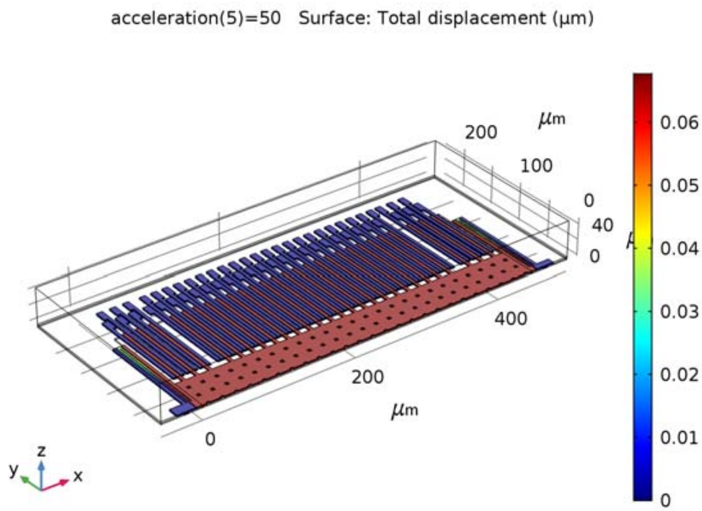


Figure 5: Displacement of the polysilicon domains when the applied acceleration is 50 g. The proof mass moves by about 0.07 micrometer. The anchored spring bases and the fixed electrodes have very little movement. The springs have varying displacement along its length as expected.

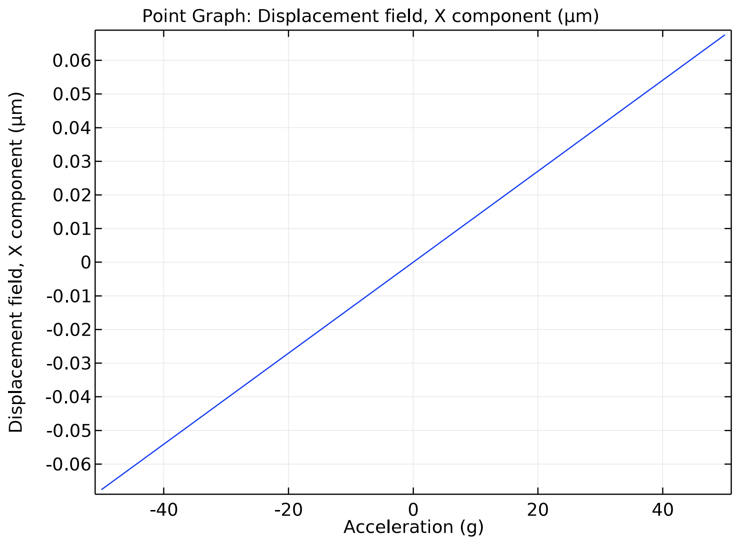


Figure 6: Displacement vs acceleration.

Figure 6 shows the linear relationship between the displacement and the applied acceleration. The displacement is measured via the capacitive coupling between the moving and the fixed sense electrodes. In the real device, during normal operation, the proof mass with its attached moving electrodes is floating at a potential close to one half of the supply voltage, and a high frequency square wave swinging between zero and the full supply voltage is applied with opposite phase to the fixed sense electrodes on each side of the moving electrodes. The fixed self test electrodes are biased at one half of the supply voltage. When the proof mass moves as a result of the acceleration, an alternating voltage in proportion to the displacement is induced due to the capacitive coupling between the fixed and moving electrodes. This arrangement nulls the average electrostatic force between the fixed and moving sense electrodes, and facilitates easier signal processing in the attached circuitry. In this example the stationary part of the square wave is modeled using a stationary study, so that the problem solves relatively quickly. The bias is shifted to zero for convenience, and the amplitude of the square wave is divided by an artificial factor of 1000 to reduce the electrostatic force between the fixed and moving sense electrodes (in practice the time average of the force will be zero due to the high frequency excitation). For a 5 V supply in the physical device, this corresponds to applying a +/-2.5 mV on the right-side and left-side fixed sense electrodes in the model. In post processing the artificial factor of 1000 is multiplied back to the sensed voltage of the proof mass. Figure 7 shows the linear relationship between the sense voltage and the acceleration. This signal is fed into an amplifier that in the real device was built on the same substrate as the mechanical structure.

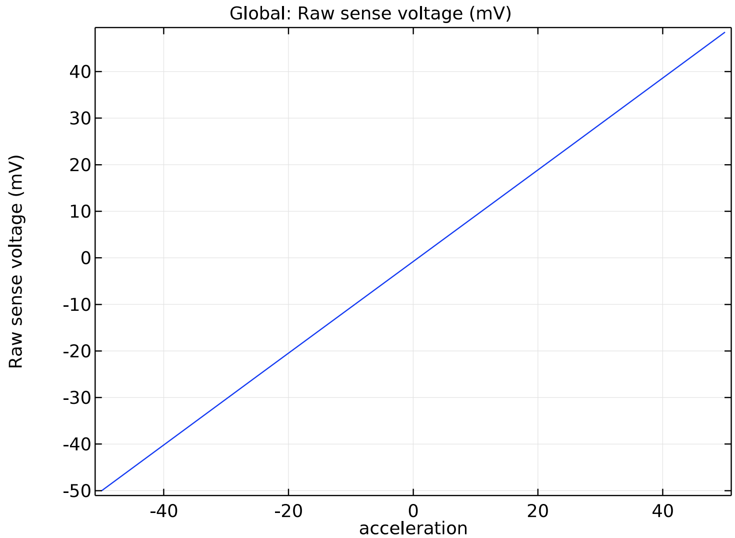


Figure 7: Sensed voltage vs. applied acceleration.

The accelerometer in the model was designed with self test electrodes that could be employed to calibrate the device in the factory. The second study illustrates the self testing by applying a bias of 2 V on the fixed self test electrodes, which are at the side of the moving electrodes attached to the proof mass. The electric field between the fixed and the moving electrodes exerts an electrostatic force that causes the proof mass to move. Figure 8 shows the displacement of the polysilicon domains when 0 V is applied to the fixed self test electrodes on the left-hand side of the moving electrodes attached to the proof mass, and 2 V to those on the right-hand side. The proof mass moves by about 0.02 μm, which is large enough in magnitude for the self test purpose (compared to the 0.07 μm of full range displacement shown in Figure 5 and Figure 6).

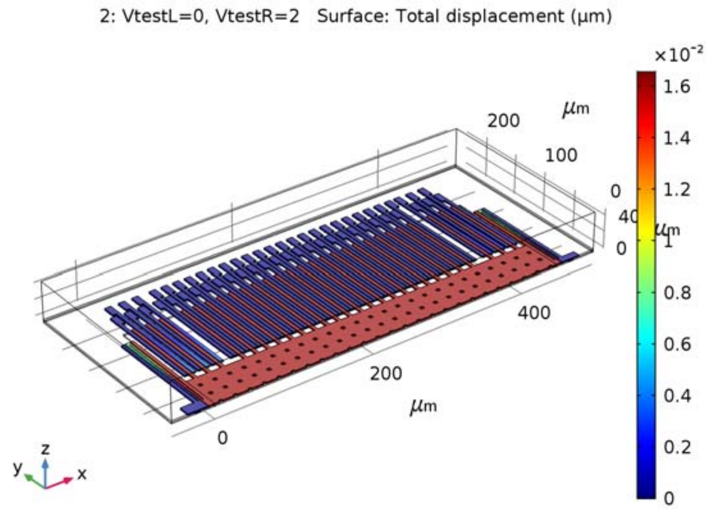


Figure 8: Displacement of the polysilicon domains when 0 V is applied to the fixed self test electrodes on the left-hand side of the moving electrodes attached to the proof mass, and 2 V to those on the right-hand side. The proof mass moved by about 0.02 µ*m, which is large enough in magnitude for the self test purpose (compared to the 0.07* µ*m of full range displacement shown in* Figure 5 *and* Figure 6*).*

Figure 9 compares the displacement obtained from applying the self test voltage to each side of the fixed electrodes. The displacement values have the same magnitude with opposite signs, as expected from symmetry.

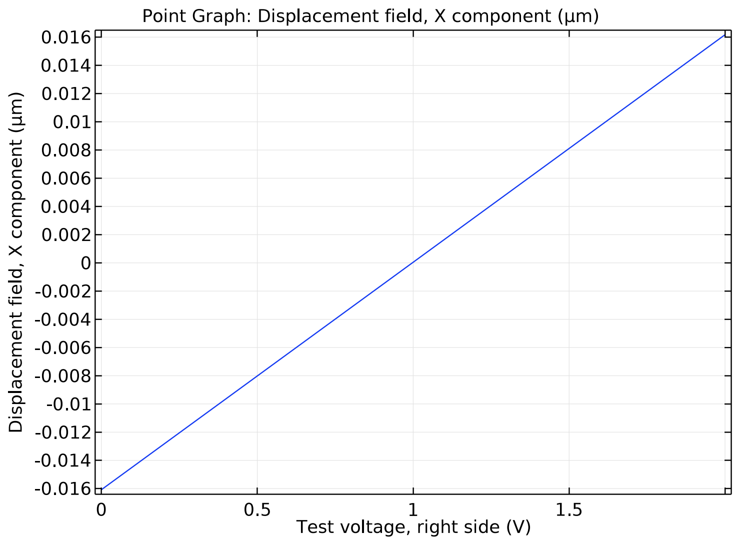


Figure 9: Displacement vs. applied self test voltage.

References

1. S.D. Senturia, *Microsystem Design* 5th ed., Kluwer Academic Publishers, pp. 513–525, 2003.

Application Library path: `MEMS_Module/Sensors/surface_micromachined_accelerometer`

Modeling Instructions

Load the geometry file.

ROOT

1 From the **File** menu, choose **Open**.

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

The model geometry has been set up using parts in a linked file. It is easier to visualize with wireframe rendering.

COMPONENT 1 (COMP1)

1 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

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

ADD MATERIAL

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

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

3 In the tree, select **Built-In>Air**.

4 Click **Add to Component 1**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **MEMS>Semiconductors>Si - Polycrystalline Silicon**.

3 Click **Add to Component 1**.

MATERIALS

Si - Polycrystalline Silicon (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Si - Polycrystalline Silicon (mat2)**.

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

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

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

ELECTROMECHANICS (EMI)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromechanics (emi)** 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 **Polysilicon**.

4 In the **Model Builder** window, collapse the **Electromechanics (emi)** node.

5 In the **Model Builder** window, expand the **Electromechanics (emi)** node.

Body Load 1

1 Right-click **Electromechanics (emi)** and choose the domain setting **Structural>Body Load**.

2 In the **Settings** window for **Body Load**, locate the **Force** section.

3 Specify the $\mathbf{F}_V$ vector as

`acceleration*emi.rho*g_const`	x
`0`	y
`0`	z

4 Locate the **Domain Selection** section. From the **Selection** list, choose **Polysilicon**.

Symmetry 1

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Structural>Symmetry**.

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

3 From the **Selection** list, choose **Symmetry plane**.

Fixed Constraint 1

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Structural>Fixed Constraint**.

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

3 From the **Selection** list, choose **Anchor plane**.

Ground 1

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Electrical>Ground**.

2 Select Boundary 45 only.

Terminal 1

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Electrical>Terminal**.

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

3 From the **Terminal type** list, choose **Voltage**.

4 In the V_0 text field, type `-2.5[mV]`.

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

6 Locate the **Boundary Selection** section. From the **Selection** list, choose **Sense left boundaries**.
7 Right-click **Component 1 (comp1)>Electromechanics (emi)>Terminal 1** and choose **Rename**.
8 In the **Rename Terminal** dialog box, type `Sense Terminal L` in the **New label** text field.
9 Click **OK**.

Terminal 2

1 In the **Model Builder** window, right-click **Electromechanics (emi)** and choose the boundary condition **Electrical>Terminal**.
2 In the **Settings** window for **Terminal**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Sense right boundaries**.
4 Locate the **Terminal** section. From the **Terminal type** list, choose **Voltage**.
5 In the V_0 text field, type `2.5[mV]`.
6 Right-click **Component 1 (comp1)>Electromechanics (emi)>Terminal 2** and choose **Rename**.
7 In the **Rename Terminal** dialog box, type `Sense Terminal R` in the **New label** text field.
8 Click **OK**.

Floating Potential 1

1 In the **Model Builder** window, right-click **Electromechanics (emi)** and choose the boundary condition **Electrical>Floating Potential**.
2 In the **Settings** window for **Floating Potential**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Proof mass boundaries**.

Terminal 3

1 Right-click **Electromechanics (emi)** and choose the boundary condition **Electrical>Terminal**.
2 In the **Settings** window for **Terminal**, locate the **Terminal** section.
3 From the **Terminal type** list, choose **Voltage**.
4 In the V_0 text field, type `VtestL`.
5 Right-click **Component 1 (comp1)>Electromechanics (emi)>Terminal 3** and choose **Rename**.
6 In the **Rename Terminal** dialog box, type `Self Test Terminal L` in the **New label** text field.
7 Click **OK**.

8 In the **Settings** window for **Terminal**, locate the **Boundary Selection** section.

9 From the **Selection** list, choose **Self test left boundaries**.

Terminal 4

1 In the **Model Builder** window, right-click **Electromechanics (emi)** and choose the boundary condition **Electrical>Terminal**.

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

3 From the **Selection** list, choose **Self test right boundaries**.

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

5 In the V_0 text field, type `VtestR`.

6 Right-click **Component 1 (comp1)>Electromechanics (emi)>Terminal 4** and choose **Rename**.

7 In the **Rename Terminal** dialog box, type `Self Test Terminal R` in the **New label** text field.

8 Click **OK**.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Meshing plane**.

4 Click **Build Selected**.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 Right-click **Swept 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.

4 Click **Clear Selection**.

5 Select Domain 3 only.

6 Click **Build All**.

STUDY 1

1 In the **Model Builder** window, right-click **Study 1** and choose **Rename**.

2 In the **Rename Study** dialog box, type `Study 1: Normal Operation` in the **New label** text field.

3 Click **OK**.

STUDY 1: NORMAL OPERATION

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1: Normal Operation** 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**.

5 Click **Range**.

6 In the **Range** dialog box, type `-50` in the **Start** text field.

7 In the **Step** text field, type `25`.

8 In the **Stop** text field, type `50`.

9 Click **Add**.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1: Normal Operation>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Segregated 1** node.

4 In the **Model Builder** window, expand the **Study 1: Normal Operation>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Iterative 1** node, then click **Study 1: Normal Operation>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Segregated 1>Segregated Step 1**.

5 In the **Settings** window for **Segregated Step**, locate the **General** section.

6 From the **Linear solver** list, choose **Iterative 1**.

7 In the **Model Builder** window, under **Study 1: Normal Operation>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Iterative 1** click **Multigrid 1**.

8 In the **Settings** window for **Multigrid**, locate the **General** section.

9 From the **Solver** list, choose **Geometric multigrid**.

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

RESULTS

Point Graph 1

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

2 In the **Model Builder** window, right-click **1D Plot Group 3** and choose **Point Graph**.

3 In the **Settings** window for **Point Graph**, locate the **Selection** section.

4 Select the **Active** toggle button.

5 Select Point 65 only.

6 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Electromechanics (Solid Mechanics)>Displacement>Displacement field (material and geometry frames)>u - Displacement field, X component**.

1D Plot Group 3

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 3** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Displacement vs. Acceleration` in the **New label** text field.

3 Click **OK**.

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

5 Select the **x-axis label** check box.

6 In the associated text field, type `Acceleration (g)`.

7 On the **Displacement vs. Acceleration** toolbar, click **Plot**.

Global 1

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

2 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Global**.

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

4 In the table, enter the following settings:

Expression	Unit	Description
emi.V0_fp1*1000	mV	Raw sense voltage

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

6 On the **1D Plot Group 4** toolbar, click **Plot**.

1D Plot Group 4

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 4** and choose **Rename**.
2 In the **Rename 1D Plot Group** dialog box, type `Sense V vs. Acceleration` in the **New label** text field.
3 Click **OK**.

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**.

STUDY 2

Step 1: Stationary

1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.
2 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.
3 In the **Settings** window for **Stationary**, locate the **Study Extensions** section.
4 Select the **Auxiliary sweep** check box.
5 Click **Add**.
6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
VtestL	2 0	

7 Click **Add**.
8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
VtestR	0 2	

9 From the **Run continuation for** list, choose **No parameter**.
10 In the **Model Builder** window, right-click **Study 2** and choose **Rename**.
11 In the **Rename Study** dialog box, type `Study 2: Self Test` in the **New label** text field.
12 Click **OK**.

STUDY 2: SELF TEST

Solution 2 (sol2)

1 On the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.
3 In the **Model Builder** window, expand the **Study 2: Self Test>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1>Segregated 1** node.
4 In the **Model Builder** window, expand the **Study 2: Self Test>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1>Iterative 1** node, then click **Study 2: Self Test>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1>Segregated 1>Segregated Step 1**.
5 In the **Settings** window for **Segregated Step**, locate the **General** section.
6 From the **Linear solver** list, choose **Iterative 1**.
7 In the **Model Builder** window, under **Study 2: Self Test>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1>Iterative 1** click **Multigrid 1**.
8 In the **Settings** window for **Multigrid**, locate the **General** section.
9 From the **Solver** list, choose **Geometric multigrid**.
10 On the **Study** toolbar, click **Compute**.

RESULTS

Displacement (emi) 1

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

Slice 1

1 In the **Model Builder** window, expand the **Potential (emi) 1** node, then click **Slice 1**.
2 In the **Settings** window for **Slice**, locate the **Plane Data** section.
3 In the **Planes** text field, type 8.
4 On the **Potential (emi) 1** toolbar, click **Plot**.

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 2: Self Test/Solution 2 (sol2)**.

Point Graph 1

1 Right-click **1D Plot Group 7** and choose **Point Graph**.

2 Select Point 65 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1> Electromechanics (Solid Mechanics)>Displacement> Displacement field (material and geometry frames)>u - Displacement field, X component**.

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

5 In the **Expression** text field, type `VtestR`.

6 On the **1D Plot Group 7** toolbar, click **Plot**.

1D Plot Group 7

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 7** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Displacement vs. Self Test V` in the **New label** text field.

3 Click **OK**.

Appendix — Geometry 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**.

GLOBAL DEFINITIONS

Geometry Parts

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Geometry Parts**.

2 On the **Geometry** toolbar, click **Parts** and choose **Load Part**.

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

4 In the **Load Part** dialog box, In the **Select parts** list, choose **Proof mass with fingers**, **Spring and anchor**, and **Electrode array**.

5 Click **OK**.

ELECTROMECHANICS (EMI)

In the **Model Builder** window, collapse the **Component 1 (comp1)>Electromechanics (emi)** node.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `surface_micromachined_accelerometer_parameters.txt`.

GEOMETRY 1

Proof mass with fingers 1 (pi1)

1 On the **Geometry** toolbar, click **Parts** and choose **Proof mass with fingers**.

2 In the **Settings** window for **Part Instance**, type `Part Link: Proof mass` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
l_PM	`l_PM`	4.48E-4 m	Proof mass length
w_PM	`w_PM`	1E-4 m	Proof mass full width
t_PM	`tSi`	2E-6 m	Proof mass thickness
l_f	`l_f`	1.14E-4 m	Length of finger
w_f	`w_f`	4E-6 m	Width of finger
n_st	`n_st`	3	Number of self test fingers
n_f	`n_f`	21	Number of sense fingers
g_f	`g_f`	1E-6 m	Gap between sense fingers
g_st	`g_st`	3E-6 m	Gap between self test fingers

Name	Expression	Value	Description
x_st	`x_st`	1E-5 m	Starting position of self test fingers
x_f	`x_f`	7.2E-5 m	Starting position of sense fingers
w_eh	`w_eh`	4E-6 m	Etch hole size
p_eh	`p_eh`	1.8E-5 m	Etch hole period

4 Click **Build All Objects**.

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

Spring and anchor 1 (pi2)

1 On the **Geometry** toolbar, click **Parts** and choose **Spring and anchor**.

2 In the **Settings** window for **Part Instance**, type `Part Link: Spring 1` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
l_sp	`l_sp`	2.8E-4 m	Spring length
w_sp	`w_sp`	2E-6 m	Spring width
g_sp	`g_sp`	1E-6 m	Spring gap
w_sp_conn	`w_sp_conn`	4E-6 m	Spring connection width
w_f	`w_f`	4E-6 m	Guard finger width
l_anch_base	`l_anch_base`	1.7E-5 m	Anchor base length
w_anch_base	`w_anch_base`	1.7E-5 m	Anchor base width
r_anch	`r_anch`	3E-6 m	Anchor radius
x_anch	`x_anch`	1.2E-5 m	Anchor position
t_sp	`tSi`	2E-6 m	Spring thickness
t_anch	`tOx`	1.6E-6 m	Anchor thickness
x_sp	`l_PM`	4.48E-4 m	Position

4 Click **Build All Objects**.

Part Link: Spring 1.1 (pi3)

1 Right-click **Part Link: Spring 1** and choose **Duplicate**.

2 In the **Settings** window for **Part Instance**, type `Part Link: Spring 2` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
mirror	0	0	0: no mirror. 1: mirror
l_sp	`l_sp+10[um]`	2.9E-4 m	Spring length
w_anch_base	`w_anch_base+10[um]`	2.7E-5 m	Anchor base width
x_sp	`0[um]`	0 m	Position

4 Click **Build All Objects**.

Electrode array 1 (pi4)

1 On the **Geometry** toolbar, click **Parts** and choose **Electrode array**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
LH	0	0	0: RH, 1: LH
l_e	`l_e_l`	1.4E-4 m	Electrode length
w_e	`w_f`	4E-6 m	Electrode width
l_p	`l_p`	1.6E-5 m	Pad length
w_p	`w_p`	8E-6 m	Pad width
r_an	`r_an`	3E-6 m	Anchor radius
t_e	`tSi`	2E-6 m	Electrode thickness
t_an	`tOx`	1.6E-6 m	Anchor thickness
n_e	`n_f+1`	22	Number of electrodes
p_e	`3*(w_f+g_f)`	1.5E-5 m	Periodicity
x_e	`x_f-w_f-g_f`	6.7E-5 m	x position
y_e	`w_PM/2+l_f-l_ovrlp`	6E-5 m	y position

4 In the **Label** text field, type `Part Link: Sense Electrodes L`.

5 Click **Build All Objects**.

6 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

Part Link: Sense Electrodes L 1 (pi5)

1 Right-click **Part Link: Sense Electrodes L** and choose **Duplicate**.

2 In the **Settings** window for **Part Instance**, type `Part Link: Sense Electrodes R` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
LH	1	1	0: RH, 1: LH
l_e	l_e_s	1.2E-4 m	Electrode length
x_e	x_f-2*(w_f+g_f)	62 µm	x position

4 Click **Build All Objects**.

Part Link: Sense Electrodes R 1 (pi6)

1 Right-click **Component 1 (comp1)>Geometry 1>Part Link: Sense Electrodes R** and choose **Duplicate**.

2 In the **Settings** window for **Part Instance**, type `Part Link: Self Test Electrodes L 1` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
LH	0	0	0: RH, 1: LH
n_e	n_st	3	Number of electrodes
p_e	3*w_f+2*g_f+g_st	1.7E-5 m	Periodicity
x_e	x_f-2*(w_f+g_f)	62 µm	x position

4 Click **Build All Objects**.

5 In the table, enter the following settings:

Name	Expression	Value	Description
x_e	x_st-w_f-g_f	5E-6 m	x position

6 Click **Build All Objects**.

Part Link: Self Test Electrodes L 1.1 (pi7)

1 Right-click **Component 1 (comp1)>Geometry 1>Part Link: Self Test Electrodes L 1** and choose **Duplicate**.

2 In the **Settings** window for **Part Instance**, type `Part Link: Self Test Electrodes L 2` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
l_e	`l_e_l`	140 µm	Electrode length
x_e	`l_PM-(x_st+w_f+g_f)-(n_st-1)*(3*w_f+2*g_f+g_st)-w_f`	3.95E-4 m	x position

4 Click **Build All Objects**.

Part Link: Self Test Electrodes L 2.1 (pi8)

1 Right-click **Component 1 (comp1)>Geometry 1>Part Link: Self Test Electrodes L 2** and choose **Duplicate**.

2 In the **Settings** window for **Part Instance**, type `Part Link: Self Test Electrodes R 1` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
LH	`1`	1	0: RH, 1: LH
x_e	`x_st-w_f-g_f+2*(w_f+g_f)`	1.5E-5 m	x position

4 Click **Build All Objects**.

Part Link: Self Test Electrodes R 1.1 (pi9)

1 Right-click **Component 1 (comp1)>Geometry 1>Part Link: Self Test Electrodes R 1** and choose **Duplicate**.

2 In the **Settings** window for **Part Instance**, type `Part Link: Self Test Electrodes R 2` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
l_e	`l_e_s`	120 µm	Electrode length
x_e	`l_PM-(x_st+w_f+g_f)-(n_st-1)*(3*w_f+2*g_f+g_st)-w_f+2*(w_f+g_f)`	4.05E-4 m	x position

4 Click **Build All Objects**.

5 Click **Go to Default View**.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Air box` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `l_polySi+40[um]`.

4 In the **Depth** text field, type `hw_polySi+20[um]`.

5 In the **Height** text field, type `50[um]`.

6 Locate the **Position** section. In the **x** text field, type `-l_spAssm-20[um]`.

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

Air box (blk1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Air box (blk1)**.

2 In the **Settings** window for **Block**, click to expand the **Layers** section.

3 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	tOx
Layer 2	tSi

4 Click **Build All Objects**.

5 In the **Model Builder** window, click **Geometry 1**.

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

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

8 In the **Model Builder** window, click **Air box (blk1)**.

9 In the **Settings** window for **Block**, click **Build All Objects**.

10 Click **Go to Default View**.

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, type `Ground plane` in the **Label** text field.

3 Locate the **Plane Definition** section. In the **z-coordinate** text field, type `-tOx`.

Plane Geometry

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>Ground plane (wp1)** click **Plane Geometry**.

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

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

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

4 In the **Height** text field, type `w_PM/2+l_f`.

5 Click **Build Selected**.

6 In the **Model Builder** window, click **Geometry 1**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

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

3 In the table, enter the following settings:

Distances (µm)
tOx

4 Click **Build Selected**.

Part Link: Proof mass (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Part Link: Proof mass (pi1)**.

2 In the **Settings** window for **Part Instance**, click to expand the **Domain selections** section.

3 Locate the **Domain Selections** section. Click to select row number 1 in the table.

4 Click **New Cumulative Selection**.

5 In the **New Cumulative Selection** dialog box, type `Polysilicon` in the **Name** text field.

6 Click **OK**.

7 In the **Settings** window for **Part Instance**, click to expand the **Boundary selections** section.

8 Locate the **Boundary Selections** section. Click **New Cumulative Selection**.

9 In the **New Cumulative Selection** dialog box, type `Proof mass boundaries` in the **Name** text field.

10 Click **OK**.

11 In the **Settings** window for **Part Instance**, locate the **Boundary Selections** section.

12 Click to select row number 1 in the table.

13 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Proof Mass + Fingers Seq	csel2		√

Part Link: Spring 1 (pi2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Part Link: Spring 1 (pi2)**.

2 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

3 Click to select row number 1 in the table.

4 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Spring + anchor	csel1		√

5 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Spring + anchor	csel2		√

Part Link: Spring 2 (pi3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Part Link: Spring 2 (pi3)**.

2 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

3 Click to select row number 1 in the table.

4 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Spring + anchor	csel1		√

5 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Spring + anchor	csel2		√

Part Link: Sense Electrodes L (pi4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Part Link: Sense Electrodes L (pi4)**.

2 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

3 Click to select row number 1 in the table.

4 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Electrode array	csel1		√

5 Click **New Cumulative Selection**.

6 In the **New Cumulative Selection** dialog box, type `Sense left boundaries` in the **Name** text field.

7 Click **OK**.

Part Link: Sense Electrodes R (pi5)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Part Link: Sense Electrodes R (pi5)**.

2 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

3 Click to select row number 1 in the table.

4 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Electrode array	csel1		√

5 Click **New Cumulative Selection**.

6 In the **New Cumulative Selection** dialog box, type `Sense right boundaries` in the **Name** text field.

7 Click **OK**.

Part Link: Self Test Electrodes L 1 (pi6)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Part Link: Self Test Electrodes L 1 (pi6)**.

2 In the **Settings** window for **Part Instance**, locate the **Boundary Selections** section.

3 Click **New Cumulative Selection**.

4 In the **New Cumulative Selection** dialog box, type `Self test left boundaries` in the **Name** text field.

5 Click **OK**.

6 In the **Settings** window for **Part Instance**, locate the **Boundary Selections** section.

7 Click to select row number 1 in the table.

8 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Electrode array	csel5		√

9 In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1>Cumulative Selections** node.

GEOMETRY 1

Part Link: Self Test Electrodes L 2 (pi7)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node, then click **Component 1 (comp1)>Geometry 1>Part Link: Self Test Electrodes L 2 (pi7)**.

2 In the **Settings** window for **Part Instance**, locate the **Boundary Selections** section.

3 Click to select row number 1 in the table.

4 Locate the **Domain Selections** section. Click to select row number 1 in the table.

5 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Electrode array	csel1		√

6 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Electrode array	csel5		√

Part Link: Self Test Electrodes R 1 (pi8)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Part Link: Self Test Electrodes R 1 (pi8)**.

2 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

3 Click to select row number 1 in the table.

4 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Electrode array	csel1		√

5 Locate the **Boundary Selections** section. Click **New Cumulative Selection**.

6 In the **New Cumulative Selection** dialog box, type `Self test right boundaries` in the **Name** text field.

7 Click **OK**.

8 In the **Settings** window for **Part Instance**, locate the **Boundary Selections** section.

9 Click to select row number 1 in the table.

10 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Electrode array	csel6		√

Part Link: Self Test Electrodes R 2 (pi9)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Part Link: Self Test Electrodes R 2 (pi9)**.

2 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

3 Click to select row number 1 in the table.

4 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Electrode array	csel1		√

5 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Electrode array	csel6		√

Part Link: Self Test Electrodes L 1 (pi6)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Part Link: Self Test Electrodes L 1 (pi6)**.

2 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

3 Click to select row number 1 in the table.

4 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Electrode array	csel1		√

Ground plane (wp1)

1 In the **Model Builder** window, collapse the **Component 1 (comp1)>Geometry 1>Ground plane (wp1)** node.

2 In the **Model Builder** window, collapse the **Component 1 (comp1)>Geometry 1>Cumulative Selections** node.

GEOMETRY 1

In the **Model Builder** window, collapse the **Component 1 (comp1)>Geometry 1** node.

DEFINITIONS

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 **y minimum** text field, type `-0.1`.

5 In the **y maximum** text field, type `0.1`.

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

7 In the **Label** text field, type `Symmetry plane`.

Box 2

1 On the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for **Box**, type `Anchor plane` 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 **z minimum** text field, type `-t0x*1.01`.

5 In the **z maximum** text field, type `-t0x*0.99`.

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

Box 3

1 On the **Definitions** toolbar, click **Box**.

2 In the **Settings** window for **Box**, type `Meshing plane` 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 **z minimum** text field, type `-t0x*0.01`.

5 In the **z maximum** text field, type `t0x*0.01`.

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

Created in COMSOL Multiphysics 5.3

Thermoelastic Damping in a MEMS Resonator

Introduction

High quality factor MEMS resonators are the key components in the emerging MEMS timing industry. In these applications a MEMS resonator is driven at its resonant frequency by a feedback loop to produce a circuit that oscillates at a fixed frequency. Such frequency references are used in a huge range of electronic devices, from CPU clocks to mobile phones. For oscillator applications the quality factor of the resonator, together with the stability of the resonant frequency, determines the ultimate performance achievable. Higher quality factor resonators have a sharper peak in their frequency spectrum at the resonant frequency and therefore pick out a particular frequency with higher fidelity. For many resonant modes the limit to the achievable quality factor is determined by thermoelastic damping.

To understand thermoelastic damping consider the stretching of a thermally isolated elastic rod. When such a rod is stretched uniformly and reversibly its temperature drops. The drop in temperature compensates for the increase in entropy caused by the stress in the rod (since the process is reversible the entropy remains constant). Similarly on compression the rod heats up. When a structure vibrates in a more complex normal mode there are some regions of compression and some of extension. Depending on the timescale of the vibration heat flows from the warmer parts of the structure to the cooler parts. Since heat flow is an irreversible process, this heat flow is associated with energy loss from the vibrational mode, and corresponding damping for the resonant mode. Thermoelastic damping is particularly important in smaller MEMS structures, in which regions of compression and expansion are in close proximity.

Model Definition

The model consists of a single beam vibrating in its fundamental mode, perpendicular to its long axis. The model geometry is shown in Figure 1. The two ends of the beam are fixed and are assumed to be connected to a much larger body (for example a contact pad) which acts as a thermal reservoir.

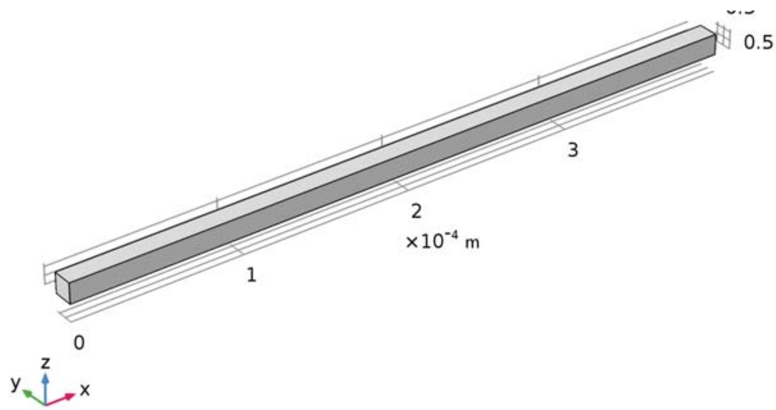


Figure 1: Symmetric model geometry. The geometry consists of a silicon beam 12 μ*m thick and 400* μ*m long. The beam width is 20* μ*m but since the geometry is symmetric only half of the beam width is shown and the symmetry boundary conditions is used.The two ends of the beam are assumed to be clamped to a body with a large thermal mass, such as a contact pad.*

This analysis computes the resonator quality factor, assuming that thermoelastic damping is the dominant damping mechanism. The coupled equations of thermoelasticity are solved within the resonator.

DERIVATION OF THE THERMOELASTICITY EQUATIONS

References 1 to 7 provide useful background information.

The equations of thermoelasticity are derived from the first law of thermodynamics, which can be stated as follows:

$$dU = dQ' + dW' \tag{1}$$

where dU is the change in internal energy, dQ' is the heat flow into the system (the prime indicates an inexact differential in this case) and dW' is the work done on the system. For a small part of a solid (sufficiently small that the stresses and strains are uniform), with an initial reference density, ρ_0, the first law can be rewritten in the following form (assuming that the differential changes occur between equilibrium states):

$$du = T_a ds + \frac{1}{\rho_0}\boldsymbol{\sigma}:d\boldsymbol{\varepsilon} \tag{2}$$

where T_a is the absolute temperature, s is the entropy per unit mass, $\boldsymbol{\sigma}$ is the elastic part of the second Piola-Kirchhoff stress (in general a rank 2 tensor), $\boldsymbol{\varepsilon}$ is the material strain (also a tensor). In general the second Piola-Kirchhoff stress tensor, $\mathbf{p}$, must be split into elastic ($\boldsymbol{\sigma}$) and inelastic ($\boldsymbol{\tau}$) parts such that:

$$\mathbf{p} = \boldsymbol{\sigma} + \boldsymbol{\tau}$$

The elastic part of the stress tensor, $\boldsymbol{\sigma}$, does work $\boldsymbol{\sigma}{:}d\boldsymbol{\varepsilon}$ during a change in the strain. The inelastic part of the strain tensor, $\boldsymbol{\tau}$, generates heat at a rate $\boldsymbol{\tau}{:}(d\boldsymbol{\varepsilon}/d\boldsymbol{\tau})$ when the strain is changing and is identified with internal or material damping. These internal damping mechanisms are associated with microscopic phenomena such as dislocation movement.

From Equation 2 it is possible to make the following identifications for T_a and $\boldsymbol{\sigma}$:

$$T_a = \left(\frac{\partial u}{\partial S}\right)_{\boldsymbol{\varepsilon}} \qquad \boldsymbol{\sigma} = \rho_0\left(\frac{\partial u}{\partial \boldsymbol{\varepsilon}}\right)_S$$

Next the *entropy balance equation* must be derived. Because thermoelasticity involves irreversible processes, the assumption of equilibrium required to derive Equation 2 is no longer valid. Instead an assumption of 'local' equilibrium is made. It is assumed that although the system is not in equilibrium, there exists within small elements a state of local equilibrium, for which the local entropy per unit mass, s, is the same function of the internal energy, strain, and particle number as it was in equilibrium. This assumption is commonly employed in the modeling of transport phenomena and is justified only by the validity of conclusions derived from it and by results obtained from specific microscopic models, for near-equilibrium situations. For a small volume element in the material frame Equation 2 can then be written as

$$\rho_0 ds = \frac{1}{T_a}\rho_0 du - \frac{1}{T_a}\boldsymbol{\sigma}{:}d\boldsymbol{\varepsilon}$$

The rate of change of entropy can then be written as

$$\rho_0\frac{ds}{dt} = \rho_0\frac{1}{T_a}\frac{du}{dt} - \frac{1}{T_a}\boldsymbol{\sigma}{:}\frac{d\boldsymbol{\varepsilon}}{dt} \tag{3}$$

From the first law (Equation 1) the rate of change of internal energy is given by:

$$\rho_0\frac{du}{dt} = \frac{dq}{dt} + \frac{dw}{dt}$$

where w is the work done per unit volume and q is the heat accumulated per unit volume. The heat accumulated can be written as the sum of the heat sources and the divergence in the material frame heat flux:

$$\frac{dq}{dt} = -(\nabla \cdot \mathbf{q}) + Q + \tau{:}\frac{d\varepsilon}{dt}$$

where Q represents the heat source per unit volume and τ is the inelastic part of the stress tensor. The rate of doing work (per unit reference volume) by a linear elastic material is given by the elastic part of the second Piola-Kirchhoff stress contracted with the rate of material strain. Per unit volume the following equation is obtained:

$$\frac{dw}{dt} = \sigma{:}\frac{d\varepsilon}{dt}$$

so Equation 3 reduces to

$$\rho_0\frac{ds}{dt} = -\frac{1}{T_a}\nabla \cdot \mathbf{q} + \frac{1}{T_a}Q + \frac{1}{T_a}\tau{:}\frac{d\varepsilon}{dt}$$

The definition of the material thermal conductivity gives

$$\mathbf{q} = -\kappa\nabla T_a$$

where κ is the thermal conductivity, defined in the material frame.

Therefore the equation is

$$T_a\rho_0\frac{ds}{dt} = \nabla \cdot (\kappa\nabla T_a) + Q + \tau{:}\frac{d\varepsilon}{dt} \tag{4}$$

It is now necessary to derive an expression for the rate of change of entropy with respect to time. In order to do this an assumption of local equilibrium is used once again. Using Equation 2 the equation is written

$$d\left(u - T_a s - \frac{1}{\rho_0}\sigma{:}\varepsilon\right) = -s\,dT_a - \frac{1}{\rho_0}\varepsilon{:}d\sigma$$

which defines a new *thermodynamic potential*, the Gibbs free energy per unit mass, given by

$$g = u - T_a s + \frac{1}{\rho_0}\sigma{:}\varepsilon$$

Changes in the Gibbs free energy per unit mass take the form

$$dg = -s\,dT_a - \frac{1}{\rho_0}\varepsilon{:}d\sigma$$

which leads to the relations

$$s = -\left(\frac{\partial g}{\partial T_a}\right)_{\boldsymbol{\sigma}} \qquad \boldsymbol{\varepsilon} = -\rho_0\left(\frac{\partial g}{\partial \boldsymbol{\sigma}}\right)_{T_a}$$

By differentiating each of the above equations a second time, it is possible to derive the following *Maxwell relation*

$$\left(\frac{\partial s}{\partial \boldsymbol{\sigma}}\right)_{T_a} = \frac{1}{\rho_0}\left(\frac{\partial \boldsymbol{\varepsilon}}{\partial T_a}\right)_{\boldsymbol{\sigma}} = -\frac{\partial^2 g}{\partial \boldsymbol{\sigma} \partial T_a} \tag{5}$$

It is now possible to derive an expression for the entropy of the solid. Assuming that the elastic stress is an invertible function of the strain, we can write $s=s(\boldsymbol{\sigma},T_a)$. Thus,

$$ds = \left(\frac{\partial s}{\partial \boldsymbol{\sigma}}\right)_{T_a} : d\boldsymbol{\sigma} + \left(\frac{\partial s}{\partial T_a}\right)_{\boldsymbol{\sigma}} dT_a$$

Using the Maxwell relation in Equation 5 gives

$$ds = \frac{1}{\rho_0}\left(\frac{\partial \boldsymbol{\varepsilon}}{\partial T_a}\right)_{\boldsymbol{\sigma}} : d\boldsymbol{\sigma} + \left(\frac{\partial s}{\partial T_a}\right)_{\boldsymbol{\sigma}} dT_a$$

so that

$$\frac{ds}{dt} = \frac{1}{\rho_0}\left(\frac{\partial \boldsymbol{\varepsilon}}{\partial T_a}\right)_{\boldsymbol{\sigma}} : \frac{d\boldsymbol{\sigma}}{dt} + \left(\frac{\partial s}{\partial T_a}\right)_{\boldsymbol{\sigma}} \frac{dT_a}{dt}$$

By definition the heat capacity of the solid at constant stress is given by

$$c_p = \left(\frac{\partial q}{\partial T}\right)_{\boldsymbol{\sigma}} = T_a\left(\frac{\partial s}{\partial T}\right)_{\boldsymbol{\sigma}}$$

Thus,

$$\frac{ds}{dt} = \frac{1}{\rho_0}\left(\frac{\partial \boldsymbol{\varepsilon}}{\partial T_a}\right)_{\boldsymbol{\sigma}} : \frac{d\boldsymbol{\sigma}}{dt} + \frac{c_p}{T_a}\frac{dT_a}{dt} \tag{6}$$

Substituting Equation 6 into Equation 4 gives the following equation for thermoelasticity:

$$\rho_0 c_p \frac{dT_a}{dt} = \nabla \cdot (\kappa \nabla T_a) + Q + \tau : \frac{d\varepsilon}{dt} - T_a\left(\frac{\partial \boldsymbol{\varepsilon}}{\partial T_a}\right)_{\boldsymbol{\sigma}} : \frac{d\boldsymbol{\sigma}}{dt} \tag{7}$$

An additional heat source term is present in Equation 7, compared to the standard heat transfer equations in solids. This term couples the structural problem with the heat transfer problem. In turn the heat transfer equation couples back into the structural problem through the constitutive relationship. COMSOL solves a linearized form of the anisotropic thermoelasticity equations given in Equation 7.

In the particular case of a linear elastic material (in the absence of damping) the stress and strain are related by Duhamel-Hooke's law:

$$(\boldsymbol{\sigma}-\boldsymbol{\sigma}_i) = \mathbf{C}:(\boldsymbol{\varepsilon}-\boldsymbol{\varepsilon}_i-\boldsymbol{\alpha}(T_a-T_{ref}))$$

where $\mathbf{C}$ is the elasticity tensor, $\boldsymbol{\sigma}_i$ is the initial stress, $\boldsymbol{\varepsilon}_i$ is the initial strain and T_{ref} is the reference temperature at which the strain and stresses take the initial values.

This equation couples the heat transfer equation to the structural problem. Given a temperature independent thermal expansivity, and no material damping, Equation 7 takes the form

$$\rho_0 c_p \frac{dT_a}{dt} = \nabla\cdot(\boldsymbol{\kappa}\nabla T_a) + Q - T_a\boldsymbol{\alpha}:\frac{d\boldsymbol{\sigma}}{dt}$$

which is the usual form of the equation for linear thermoelasticity.

Results and Discussion

Figure 2 shows the mode shape and the corresponding temperature distribution within the beam. The mode has an eigenfrequency of 63.3 kHz and a quality factor of 10700. In Ref. 4 Zener derived an approximate analytic expression for the quality factor of a thin isotropic beam vibrating in its fundamental mode, by considering only the thermal gradients in the direction of flexure. Zener's expression is given by:

$$\frac{1}{Q} = \frac{E\alpha T_a}{\rho_0 c_p}\frac{\omega\tau}{1+(\omega\tau)^2} \tag{8}$$

where E is the Young's modulus of the beam, α is the isotropic thermal expansivity, ω is the mechanical angular resonant frequency and τ is the thermal relaxation time constant of the system, given by:

$$\tau = \frac{\rho_0 c_p h^2}{\pi^2\kappa}$$

where h is the beam thickness and κ is the thermal conductivity of the mode. The resonant frequency of the beam can also be computed analytically and is given by:

$$\omega = 22{,}373\frac{h}{L^2}\sqrt{\frac{E}{12\rho_0}} \qquad (9)$$

Table 1 compares the COMSOL model with values computed using Equation 8 and Equation 9 and with experimental results, obtained from Ref. 7. Note that the COMSOL model has a slightly higher quality factor than the theoretical result, because some of the thermal gradients are removed by the isothermal boundary condition (the quality factor is reduced significantly if a thermal insulation boundary condition is applied to the end boundaries—in practice the real boundary condition is somewhere between these two extremes).

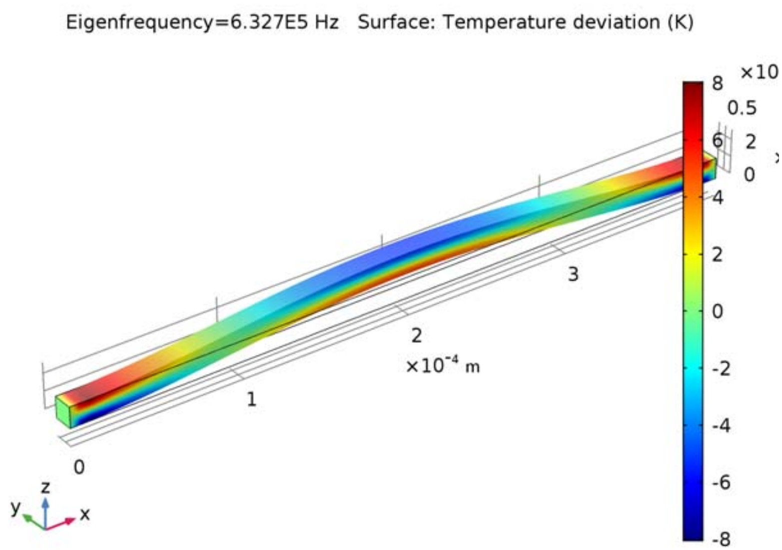


Figure 2: Fundamental mode shape and corresponding temperature distribution within the beam.

TABLE 1: COMPARISON OF RESULTS FROM THE MODEL WITH THEORY AND EXPERIMENT

SOURCE	RESONANT FREQUENCY (MHZ)	QUALITY FACTOR
COMSOL Model	0.63	10.7×10^3

TABLE 1: COMPARISON OF RESULTS FROM THE MODEL WITH THEORY AND EXPERIMENT

SOURCE	RESONANT FREQUENCY (MHZ)	QUALITY FACTOR
Equation 8 and Equation 9	0.63	10.3×10^3
Experiment (Ref. 7)	0.57	10.3×10^3

References

1. C.J. Adkins, *Equilibrium Thermodynamics*, Cambridge University Press, 1983.

2. W. Yourgrau, A. van der Merwe, and G. Raw, *Treatise on Irreversible and Statistical Thermodynamics: An Introduction to Nonclassical Thermodynamics*, Dover Publications, Inc., New York, 2002.

3. C. Zener, "Internal Friction in Solids I: Theory of Internal Friction in Reeds," *Physical Review*, vol. 52, pp. 90–99, 1937.

4. C. Zener, "Internal Friction in Solids II: General Theory of Thermoelastic Internal Friction," *Physical Review*, vol. 53, pp. 230–235, 1938.

5. M. E. Gurtin, E. Fied, and L. Anand, *The Mechanics and Thermodynamics of Continua*, Cambridge University Press, 2010.

6. V.A. Lubarda, "On Thermodynamic Potentials in Linear Thermoelasticity," *Int. J. Solids and Structures*, vol. 41, no. 26, pp. 7377–7398, 2004.

7. A. Duwel, R.N. Candler, T.W. Kenny, and M. Varghese, "Engineering MEMS Resonators with Low Thermoelastic Damping," *J. Microelectromechanical Systems*, vol. 15, no. 6, pp. 1437–1445, 2006.

Application Library path: `MEMS_Module/Actuators/thermoelastic_damping_3d`

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>Thermoelasticity (te)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Eigenfrequency**.

6 Click **Done**.

GEOMETRY 1

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

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

3 In the **Width** text field, type `4e-4`.

4 In the **Depth** text field, type `1.2e-5`.

5 In the **Height** text field, type `1.2e-5`.

6 In the **Width** text field, type `4e-4`.

7 Click **Build All Objects**.

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:

Name	Expression	Value	Description
E0	157[GPa]	1.57E11 Pa	Young's modulus
rho0	2330[kg/m^3]	2330 kg/m³	Density
nu0	0.3	0.3	Poisson's ratio
alpha0	2.6e-6[1/K]	2.6E-6 1/K	Coefficient of thermal expansion
Cp0	700[J/(kg*K)]	700 J/(kg·K)	Heat capacity
kappa0	90[W/(m*K)]	90 W/(m·K)	Thermal conductivity

MATERIALS

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	`E0`	Pa	Basic
Poisson's ratio	nu	`nu0`	1	Basic
Density	rho	`rho0`	kg/m³	Basic
Coefficient of thermal expansion	alpha	`alpha0`	1/K	Basic
Thermal conductivity	k	`kappa0`	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	`Cp0`	J/(kg·K)	Basic

THERMOELASTICITY (TE)

Symmetry 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Thermoelasticity (te)** and choose the boundary condition **Solid Mechanics>Symmetry**.

2 Select Boundary 2 only.

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Thermoelasticity (te)** and choose the boundary condition **Solid Mechanics>Fixed Constraint**.

2 Select Boundaries 1 and 6 only.

Zero Temperature Deviation 1

1 Right-click **Thermoelasticity (te)** and choose **Zero Temperature Deviation**.

2 Select Boundaries 1 and 6 only.

MESH 1

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Mapped**.

2 Select Boundary 1 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Edges 4 and 6 only.

Distribution 1

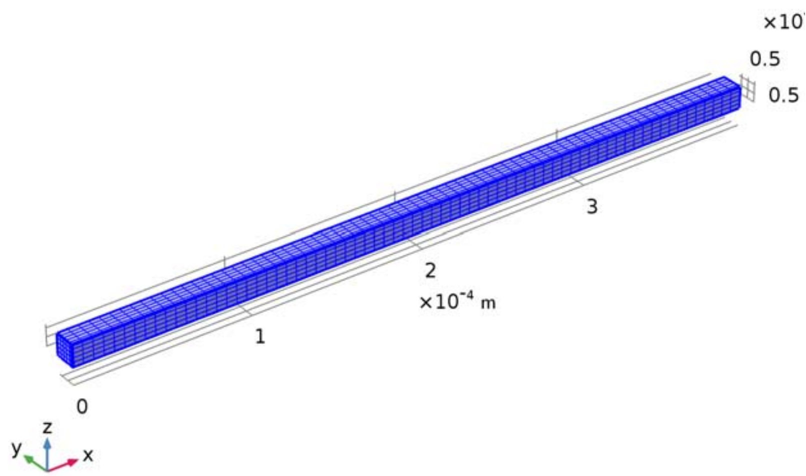
1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 Right-click **Swept 1** and choose **Distribution**.

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

4 In the **Number of elements** text field, type 70.

5 Click **Build All**.



STUDY 1

Step 1: Eigenfrequency

1. In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
2. Select the **Desired number of eigenfrequencies** check box.
3. In the associated text field, type `1`.
4. Select the **Search for eigenfrequencies around** check box.
5. In the associated text field, type `0.63e6`.
6. From the **Eigenfrequency search method around shift** list, choose **Larger real part**.
7. On the **Home** toolbar, click **Compute**.

RESULTS

Temperature Deviation (te)

1. In the **Model Builder** window, under **Results** click **Temperature Deviation (te)**.
2. On the **Temperature Deviation (te)** toolbar, click **Plot**.

   Compare the default plot with .

Global Evaluation 1

1. On 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> Thermoelasticity>Global>te.Q_eig - Quality factor for eigenvalue**.
3. Click **Evaluate**.

   Compare the result with that in .

Created in COMSOL Multiphysics 5.3

Thickness Shear Mode Quartz Oscillator

Introduction

AT cut quartz crystals are widely employed in a range of applications, from oscillators to microbalances. One of the important properties of the AT cut is that the resonant frequency of the crystal is temperature independent to first order. This is desirable in both mass sensing and timing applications. AT cut crystals vibrate in the thickness shear mode — an applied voltage across the faces of the cut produces shear stresses inside the crystal. This example considers the vibration of an AT cut thickness shear oscillator, focusing on the mechanical response of the system in the frequency domain. The effect of a series capacitor on the mechanical resonance is also considered. Adding a series capacitance is a technique frequently employed to tune crystal oscillators.

Model Definition

The model geometry is shown in Figure 1. The oscillator consists of a single (left-handed) quartz disc, supported so as not to impede the motion of the vibrational mode. There are two electrodes on the top and bottom surfaces of the geometry, one of which is grounded.

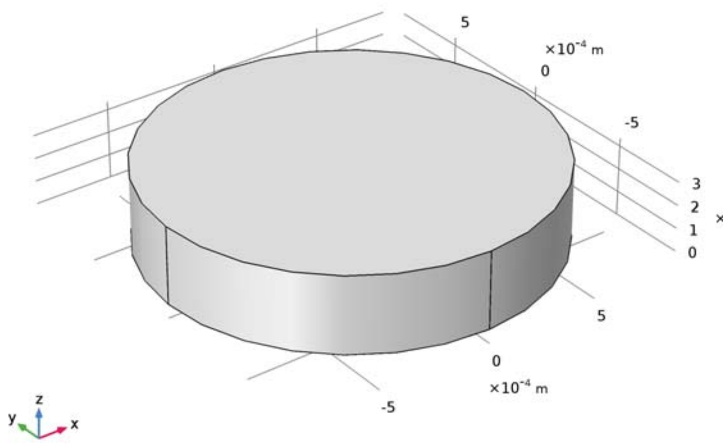


Figure 1: Model geometry.

In the first version of the model an AC voltage is applied to the top electrode. In the second version, the crystal is still driven by an AC voltage, but a capacitor is placed between the voltage source and the top electrode of the crystal.

DOMAIN LEVEL EQUATIONS

Within a piezoelectric crystal there is a coupling between the strain and the electric field, which is determined by the constitutive relation:

$$\begin{aligned} \mathbf{T} &= c_E \mathbf{S} - e^T \mathbf{E} \\ \mathbf{D} &= e \mathbf{S} + \varepsilon_S \mathbf{E} \end{aligned} \tag{1}$$

Here, $\mathbf{S}$ is the strain, $\mathbf{T}$ is the stress, $\mathbf{E}$ is the electric field, and $\mathbf{D}$ is the electric displacement field. The material parameters c_E, e, and ε_S, correspond to the material stiffness, the coupling properties, and the permittivity. These quantities are tensors of rank 4, 3, and 2 respectively, but, since the tensors are highly symmetric for physical reasons, they can be represented as matrices within an abbreviated subscript notation, which is usually more convenient. Equation 1 is implemented by the Piezoelectric Effect branch located under the Multiphysics branch. This constitutive relation is used to couple the equations of Solid Mechanics and Electrostatics, which are solved within the material.

MATERIAL ORIENTATION

The orientation of a piezoelectric crystal cut is frequently defined by the system introduced by the IRE standard of 1949 (Ref. 1). This standard has undergone a number of subsequent revisions, with the final revision being the IEEE standard of 1987 (Ref. 2). Unfortunately the 1987 standard contained a number of serious errors and the IEEE subsequently withdrew it. COMSOL Multiphysics therefore adopts the preceding 1978 standard (Ref. 3), which is similar to the 1987 standard, for material property definitions. Most of the material properties in the material library are based on the values given in the book by Auld (Ref. 4), which uses the 1978 IEEE conventions. This is consistent with general practice except in the specific case of Quartz, where it is more common to use the 1949 IRE standard to define the material properties. COMSOL Multiphysics therefore provides an additional set of material properties consistent with the 1949 standard for the case of Quartz. Note that the material properties for quartz are based on Ref. 5, which uses the 1949 IRE standard (the properties are appropriately modified according to the different standards).

The stiffness, compliance, coupling, and dielectric material property matrices are defined with the crystal axes aligned with the local coordinate axes. Note that the signs of several matrix components differ between the IRE 1949 and the IEEE 1978 standards (see Table 1). In the absence of a user-defined coordinate system, the local system corresponds to the global X, Y, and Z coordinate axes. When an alternative coordinate system is selected this system defines the orientation of the crystal axes. This is the mechanism used in COMSOL Multiphysics to define a particular crystal cut, and typically it is necessary to calculate the appropriate Euler angles for the cut (given the thickness orientation for the wafer). All piezoelectric material properties are defined using the Voigt form of the abbreviated subscript notation, which is universally employed in the literature (this differs from the standard notation used for Linear Elastic Material in the Solid Mechanics interface). The material properties are defined in the material frame, so that if the solid rotates during deformation the material properties rotate with the solid.

Crystal cuts are usually defined by a mechanism introduced by the IEEE/IRE standards. Both standards use a notation that defines the orientation of a virtual slice (the plate) through the crystal. The crystal axes are denoted X, Y, and Z and the plate, which is usually rectangular, is defined as having sides l, w, and t (length, width, and thickness). Initially the plate is aligned with respect to the crystal axes and then up to three rotations are defined, using a right-handed convention about axes embedded along the l, w, and t sides of the plate. This model uses AT cut quartz, defined in the IEEE 1978 standard as: (YXl) –35.25°. The first two letters in the bracketed expression always refer to the initial orientation of the thickness and the length of the plate. Subsequent bracketed letters then define up to three rotational axes, which move with the plate as it is rotated. Angles of rotation about these axes are specified after the bracketed expression in the order of the letters, using a right-handed convention. For AT cut quartz only one rotation, about the l axis, is required. This is illustrated in Figure 2. Note that within the 1949 IRE Standard AT cut quartz is denoted as: (YXl) +35.25°. Table 2 summarizes the differences between the standards for the AT cut.

!	When defining the material properties of Quartz, the orientation of the X, Y, and Z axes with respect to the crystal differs between the 1978 IEEE standard and the 1949 IRE standard. A consequence of this is that both the material property matrices and the crystal cuts differ between the two standards. Table 1 summarizes the signs for the important matrix elements under the two conventions. Table 2 shows the different definitions of the crystal cuts under the two conventions.

TABLE 1: SIGNS FOR THE MATERIAL PROPERTIES OF QUARTZ, WITHIN THE TWO STANDARDS COMMONLY EMPLOYED

	IRE 1949 STANDARD		IEEE 1978 STANDARD	
MATERIAL PROPERTY	RIGHT HANDED QUARTZ	LEFT HANDED QUARTZ	RIGHT HANDED QUARTZ	LEFT HANDED QUARTZ
s_{14}	+	+	-	-
c_{14}	-	-	+	+
d_{11}	-	+	+	-
d_{14}	-	+	-	+
e_{11}	-	+	+	-
e_{14}	+	-	+	-

TABLE 2: CRYSTAL CUT DEFINITIONS FOR QUARTZ CUTS WITHIN THE TWO STANDARDS COMMONLY EMPLOYED AND THE CORRESPONDING EULER ANGLES FOR DIFFERENT ORIENTATIONS OF THE CRYSTAL THICKNESS

STANDARD	REPRESENTATION	AT CUT
IRE 1949	Standard	*(YXl) +35.25°*
	Y-thickness Euler angles	*(ZXZ: 0°,–35.25°,0°)*
	Z-thickness Euler angles	*(ZXZ: 0°,–125.25°,0°)*
IEEE 1978	Standard	*(YXl) -35.25°*
	Y-thickness Euler angles	*(ZXZ: 0°, 35.25°,0°)*
	Z-thickness Euler angles	*(ZXZ: 0°,-54.75°,0°)*

When defining the material orientation it is necessary to consider the orientation of the plate with respect to the global coordinate system in addition to the orientation of the plate with respect to the crystallographic axes.

This model uses AT cut quartz, defined in the IEEE 1978 standard as shown in Figure 2. The definition of the appropriate local coordinate system depends on the desired final orientation of the plate in the global coordinate system. One way to set up the plate is to orientate its normal parallel to the Y axis in the global coordinate system. Figure 3 shows how to define the local coordinate system in this case (instructions for how to set up the crystal in this manner are provided in brackets in the step by step instructions).

Figure 4 shows how to define the local system such that the plate has its normal parallel to the global Z axis, which is the case for the crystal in this model.

Whatever crystal orientation is chosen, it is critical to keep track of the orientation of the local system with respect to the global system, which is defined depending on the desired orientation of the plate in the model.

There are also a number of methods to define the local coordinate system with respect to the global system. Usually it is most convenient to define the local coordinates with a Rotated System node, which defines three Euler angles according to the ZXZ convention (rotation about Z, then X, then Z again). Note that these Euler angles define the local (crystal) axes with respect to the global axes — this is distinct from the approach of defining the cut (global) axes with respect to the crystal (local) axes.

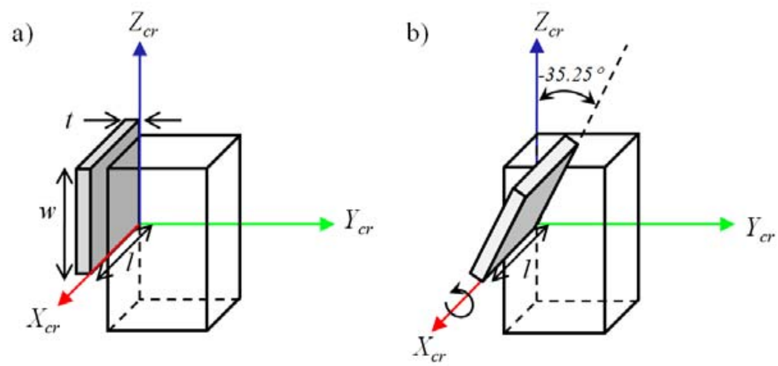


Figure 2: Definition of the AT cut of quartz within the IEEE 1978 standard. The AT cut is defined as: (YXl) −35.25°. The first two bracketed letters specify the initial orientation of the plate, with the thickness direction, t, along the crystal Y axis and the length direction, l, along the X axis. Then up to three rotations about axes that move with the plate are specified by the corresponding bracketed letters and the subsequent angles. In this case only one rotation is required about the l axis, of −35.25° (in a right-handed sense).

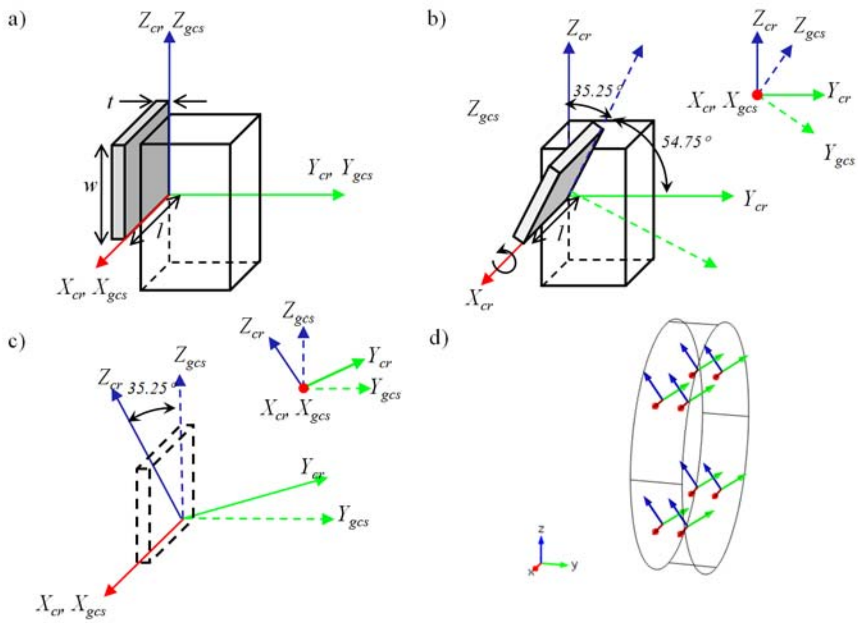


Figure 3: Defining an AT cut crystal plate within COMSOL Multiphysics, with normal in the global Y-direction. Within the 1978 IEEE standard the AT cut is defined as (YXl) -35.25°. Start with the plate normal or thickness in the Y_{cr} direction (a) and rotate the plate –35.25° about the l axis (b). The global coordinate system rotates with the plate. Finally rotate the entire system so that the global coordinate system is orientated as it appears in COMSOL Multiphysics (c). The local coordinate system should be defined with the Euler angles (ZXZ - 0, 35.25°, 0).(d) shows a coordinate system for this system in COMSOL Multiphysics.

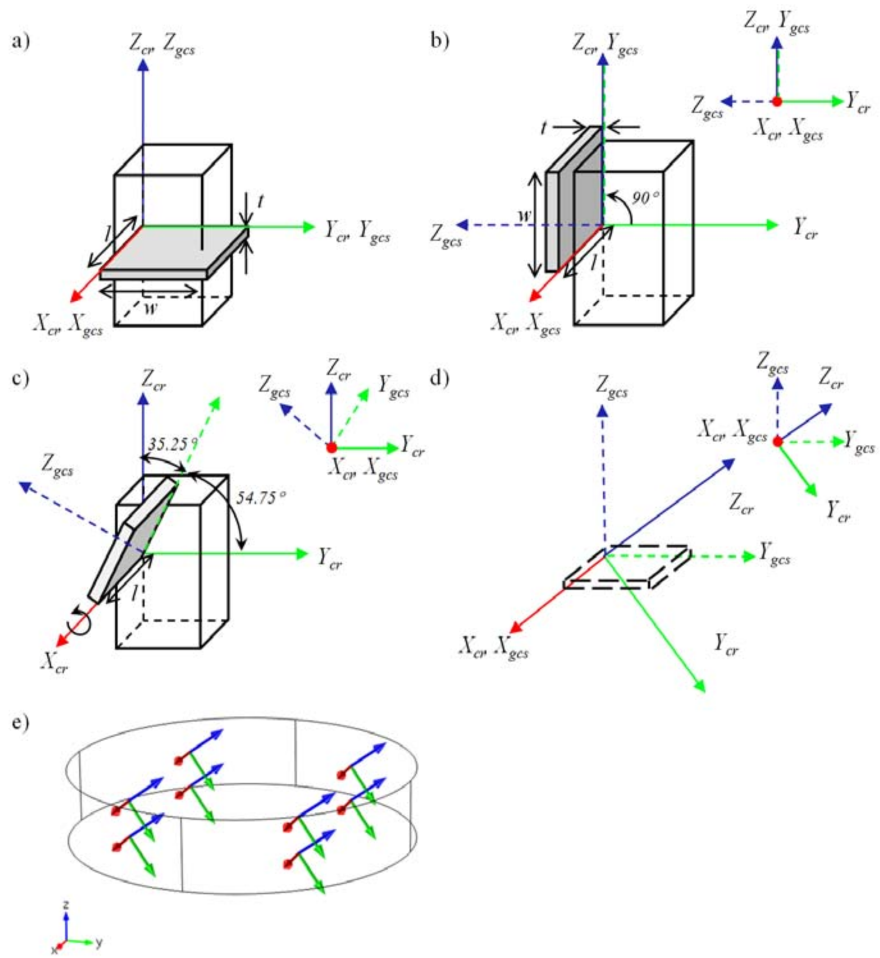


Figure 4: Defining an AT cut crystal plate within COMSOL Multiphysics, with normal in the global Z-direction. Within the 1978 IEEE standard the AT cut is defined as (YXl) – 35.25°. Begin with the plate normal in the Z_{cr}-direction, so the crystal and global systems are coincident. Rotate the plate so that its thickness points in the Y_{cr}-direction (the starting point for the IEEE definition), the global system rotates with the plate (b). Rotate the plate –35.25° about the l axis (d). Finally rotate the entire system so that the global coordinate system is orientated as it appears in COMSOL Multiphysics (d). The local coordinate system should be defined with the Euler angles (ZXZ: 0, -54.75°, 0). (e) shows a coordinate system for this system in COMSOL Multiphysics.

ELECTRICAL CIRCUIT

In the first part of the model an AC voltage is applied directly to the top plate of the oscillator, which is grounded. In the second part of the model, a capacitor is added between the voltage source and the oscillator, as shown in Figure 5. In COMSOL Multiphysics, the oscillator is coupled into the circuit using the **External I Terminal** feature. The terminal boundary condition within the model is set to **Circuit** and this feature then captures the charge generated by the circuit.

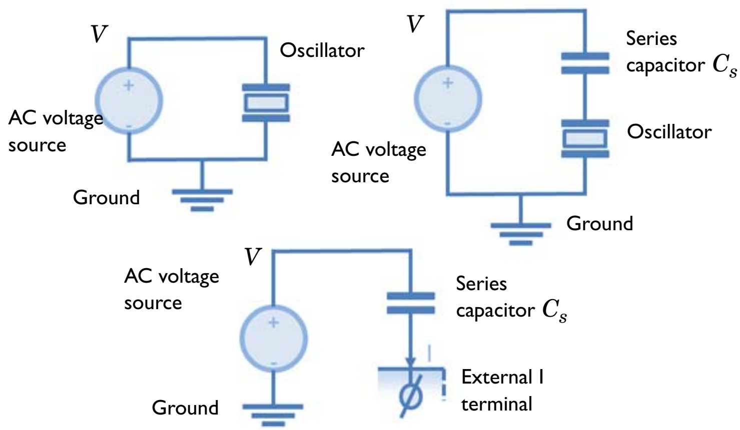


Figure 5: Top left: Electrical circuit for the first part of the model. Top right: Electrical circuit for the second part of the model. Bottom: Circuit for the second part of the model as implemented in COMSOL Multiphysics.

Results and Discussion

Figure 6 shows the crystal displacement at its resonant frequency of 5.11 MHz. The form of the displacement shows clearly the shear nature of the resonance. The potential on cut slices through the plate is illustrated in Figure 7. The mechanical domain frequency response of the oscillator is shown in Figure 8. A clear anti-resonance is apparent, with a resonant frequency close to 5.11 MHz.

The addition of a series capacitance between the oscillator and the voltage source is expected to pull the resonant frequency to higher values. Figure 9 shows that this effect occurs as expected, with the resonant frequency increasing the most for smaller values of the series capacitance (in the limit of very large series capacitance the impedance of the series capacitor goes to zero and produces the result shown in Figure 8).

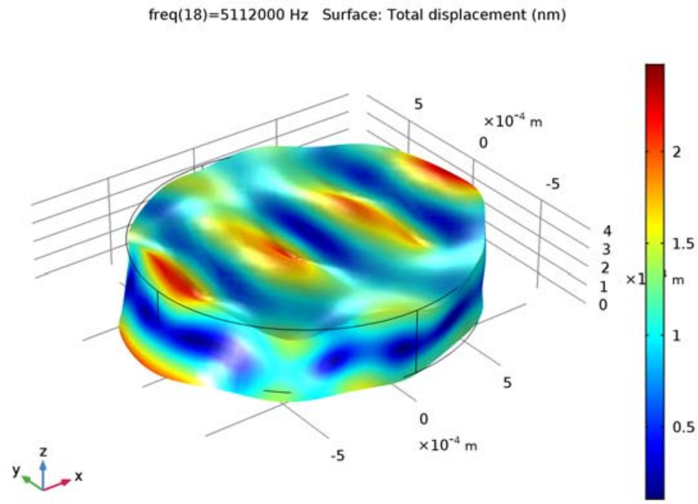


Figure 6: Displacement of the crystal at resonance.

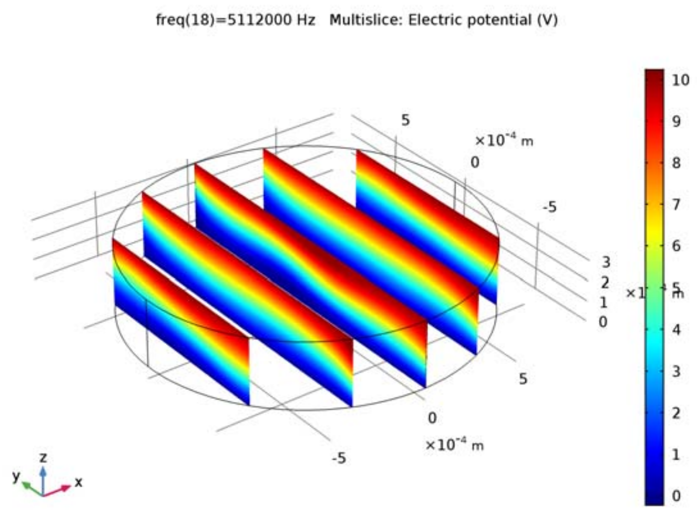


Figure 7: Electric potential inside the crystal at resonance.

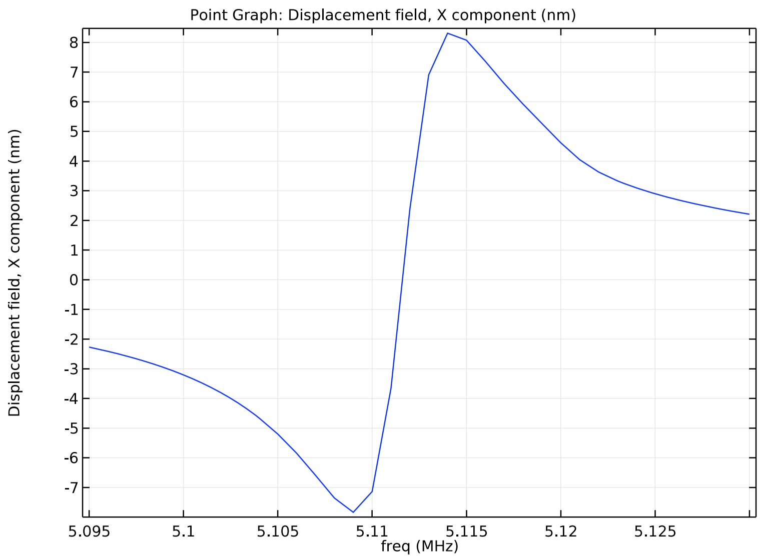


Figure 8: Mechanical response of the structure with no series capacitance.

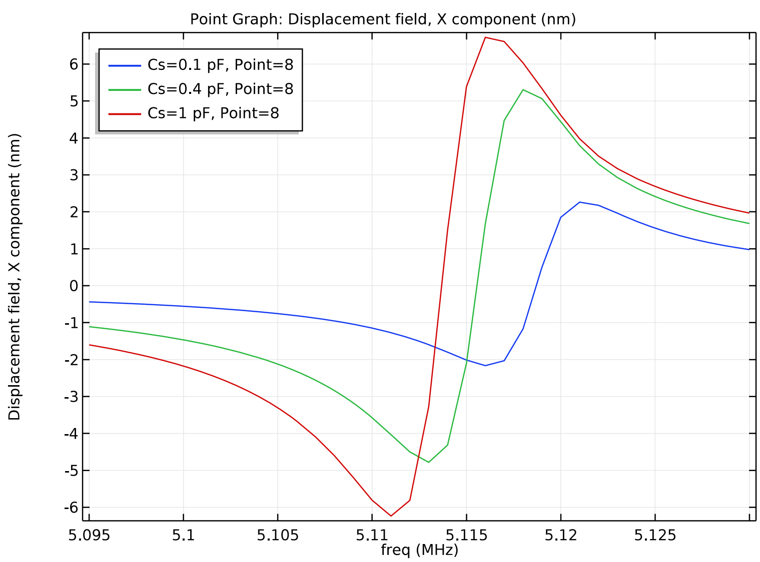


Figure 9: Mechanical response of the structure with different series capacitances.

References

1. “Standards on Piezoelectric Crystals, 1949”, *Proceedings of the I. R. E.*,vol. 37, no.12, pp. 1378–1395, 1949.

2. IEEE Standard on Piezoelectricity, ANSI/IEEE Standard 176-1987, 1987.

3. IEEE Standard on Piezoelectricity, ANSI/IEEE Standard 176-1978, 1978.

4. B. A. Auld, *Acoustic Fields and Waves in Solids*, Krieger Publishing Company, 1990.

5. R. Bechmann, “Elastic and Piezoelectric Constants of Alpha-Quartz”, *Physical Review B*, vol. 110 no. 5, pp. 1060–1061, 1958.

Application Library path: `MEMS_Module/Piezoelectric_Devices/thickness_shear_quartz_oscillator`

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>Piezoelectric Devices**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Add parameters for the model geometry and series capacitance.

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:

Name	Expression	Value	Description
Cs	1[pF]	1E-12 F	Series capacitance
R0	835[um]	8.35E-4 m	Oscillator radius
H0	334[um]	3.34E-4 m	Oscillator thickness

GEOMETRY 1

Create the geometry.

Cylinder 1 (cyl1)

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

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

3 In the **Radius** text field, type `R0`.

4 In the **Height** text field, type `H0`.

If you wish to set up the material orientation using the alternative method shown in , change the **Axis type** to *y*-axis.

5 Click **Build All Objects**.

DEFINITIONS

Set up a rotated system appropriate for AT cut Quartz.

Rotated System 2 (sys2)

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

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

3 Find the **Euler angles (Z-X-Z)** subsection. In the β text field, type `-54.75[deg]`.

If you wish to set up the material orientation using the alternative method shown in , type `35.25[deg]` in the β text field.

ADD MATERIAL

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

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

3 In the tree, select **Piezoelectric>Quartz LH (1978 IEEE)**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Quartz LH (1978 IEEE) (mat1)

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

SOLID MECHANICS (SOLID)

Use the rotated system to define the orientation of the crystal.

Piezoelectric Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Piezoelectric Material 1**.

2 In the **Settings** window for **Piezoelectric Material**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

Add damping to the model.

Mechanical Damping 1

1 Right-click **Component 1 (comp1)>Solid Mechanics (solid)>Piezoelectric Material 1** and choose **Damping**.

2 In the **Settings** window for **Mechanical Damping**, locate the **Damping Settings** section.

3 From the **Damping type** list, choose **Isotropic loss factor**.

4 From the η_s list, choose **User defined**. In the associated text field, type `1e-3`.

ELECTROSTATICS (ES)

Add electrical boundary conditions to the model. First add a **Terminal** boundary condition that connects the electrode to an external circuit.

Terminal 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electrostatics (es)** and choose the boundary condition **Terminal**.

2 Select Boundary 4 only.

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

4 From the **Terminal type** list, choose **Circuit**.

Over-ride the preceding boundary condition with a constant potential boundary condition to compute the response of the device without a series capacitance. This node will be disabled in the study when the circuit is included in the model.

Terminal 2

1 In the **Model Builder** window, right-click **Electrostatics (es)** and choose the boundary condition **Terminal**.
2 In the **Settings** window for **Terminal**, locate the **Terminal** section.
3 In the **Terminal name** text field, type 1.
4 Select Boundary 4 only.
5 From the **Terminal type** list, choose **Voltage**.
6 In the V_0 text field, type 10.

Ground 1

1 Right-click **Electrostatics (es)** and choose **Ground**.
2 Select Boundary 3 only.

ROOT

In the second study in this model, the effect of a series capacitor on the device response will be investigated. Add an **Electrical Circuit** interface to model the capacitor.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2 Go to the **Add Physics** window.
3 In the tree, select **AC/DC>Electrical Circuit (cir)**.
4 Click **Add to Component** in the window toolbar.
5 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ELECTRICAL CIRCUIT (CIR)

Features in the **Electric Circuits** interface are connected by specifying connecting node numbers for each port of the device.

Ground Node 1

A ground node is automatically added to the circuit, with the default node number of 0.

Next add a voltage source between the ground node and a (newly created) node with number 2.

Voltage Source V1

1 In the **Model Builder** window, right-click **Electrical Circuit (cir)** and choose **Voltage Source**.
2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	0

4 Locate the **Device Parameters** section. From the **Source type** list, choose **AC-source**.

5 In the V_{src} text field, type 10.

Add a capacitor between the voltage source output (node 2) and a new node, 1.

Capacitor C1

1 Right-click **Electrical Circuit (cir)** and choose **Capacitor**.

2 In the **Settings** window for **Capacitor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	1

4 Locate the **Device Parameters** section. In the C text field, type Cs.

Connect node 1 to the **Terminal** feature in the model using the **External I-Terminal** feature.

External I-Terminal 1

1 Right-click **Electrical Circuit (cir)** and choose **External Couplings>External I-Terminal**.

Couple the electric potential from the **Terminal** in the electrostatics interface back into the model.

2 In the **Settings** window for **External I-Terminal**, locate the **Node Connections** section.

3 In the **Node name** text field, type 1.

4 Locate the **External Terminal** section. From the V list, choose **Terminal voltage (es)**.

MESH 1

Create a swept triangular mesh.

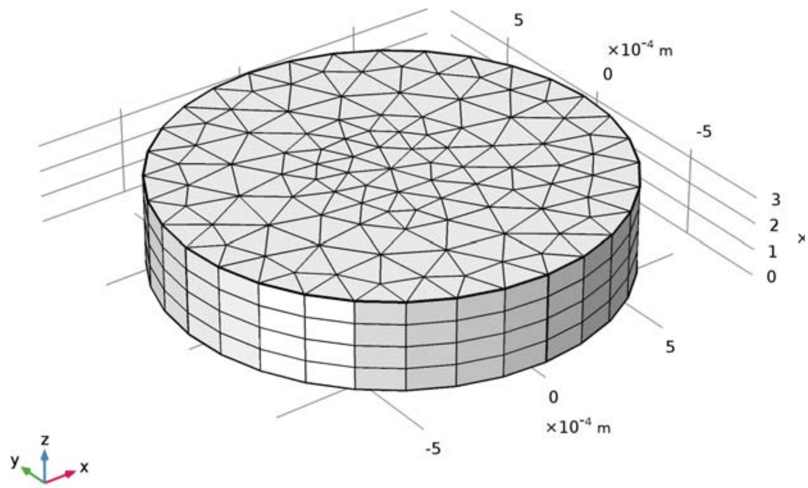
Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundary 4 only.

Distribution 1

1. In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.
2. Right-click **Swept 1** and choose **Distribution**.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type `4`.
5. Click **Build All**.



The mesh used here is somewhat coarse for the range of frequencies that are solved for. This is mainly to keep the computational time and RAM requirements as low as possible. Interested users are encouraged to solve the problem for finer mesh settings.

STUDY 1

Set up and solve a frequency dependent study.

Step 1: Frequency Domain

1. In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.
2. In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
3. Click **Range**.
4. In the **Range** dialog box, type `5.095[MHz]` in the **Start** text field.
5. In the **Step** text field, type `1[kHz]`.

6 In the **Stop** text field, type `5.13[MHz]`.

7 Click **Replace**.

In the first study, disable the electrical circuit.

8 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

9 In the table, clear the **Solve for** check box for the **Electrical Circuit (cir)** interface.

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

RESULTS

Instead of the stress plot, visualize the mode shape of the device at resonance. Note that this plot and subsequent plots will appear rotated compared to that shown in if the alternative definition of the material orientation described in is used.

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

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

3 Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **5.112E6**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Displacement** node, then click **Surface 1**.

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>Solid Mechanics> Displacement>solid.disp - Total displacement**.

3 Locate the **Expression** section. From the **Unit** list, choose **nm**.

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

5 On the **Displacement** toolbar, click **Plot**.

The second default plot shows the electric potential within the device. For a better view, plot 5 slices in *xy* planes.

Electric Potential (es)

1 In the **Model Builder** window, under **Results** click **Electric Potential (es)**.

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

3 From the **Parameter value (freq (Hz))** list, choose **5.112E6**.

Multislice 1

1 In the **Model Builder** window, expand the **Electric Potential (es)** node, then click **Multislice 1**.
2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.
3 Find the **x-planes** subsection. In the **Planes** text field, type `5`.
4 Find the **y-planes** subsection. In the **Planes** text field, type `0`.
5 Find the **z-planes** subsection. In the **Planes** text field, type `0`.
6 On the **Electric Potential (es)** toolbar, click **Plot**.

Add a plot to show the mechanical response of the device.

1D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Mechanical Response` in the **Label** text field.

Point Graph 1

1 Right-click **Mechanical Response** and choose **Point Graph**.
2 Select Point 8 only.
3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics> Displacement>Displacement field (material and geometry frames)>u - Displacement field, X component**.
4 Locate the **y-Axis Data** section. From the **Unit** list, choose **nm**.
5 Locate the **x-Axis Data** section. From the **Unit** list, choose **MHz**.
6 On the **Mechanical Response** toolbar, click **Plot**.

Now set up a study to compute the frequency response of the device with different capacitors added in series.

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> Frequency Domain**.
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: Frequency Domain

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

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog box, type `5.095[MHz]` in the **Start** text field.

5 In the **Step** text field, type `1[kHz]`.

6 In the **Stop** text field, type `5.13[MHz]`.

7 Click **Replace**.

8 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

9 Select the **Modify physics tree and variables for study step** check box.

10 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Electrostatics (es)>Terminal 2**.

11 Click **Disable**.

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Cs	`0.1 0.4 1`	pF

5 In the **Model Builder** window, click **Study 2**.

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** check box.

8 On the **Study** toolbar, click **Compute**.

RESULTS

Re-plot the mechanical response with the additional series capacitance.

Mechanical Response 1

1 In the **Model Builder** window, under **Results** right-click **Mechanical Response** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Mechanical response, Parametric` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/ Parametric Solutions 1 (sol3)**.

4 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

Point Graph 1

1 In the **Model Builder** window, expand the **Results>Mechanical response, Parametric** node, then click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

4 On the **Mechanical response, Parametric** toolbar, click **Plot**.

Note how the mechanical resonant frequency is 'pulled' by the series capacitance.

Created in COMSOL Multiphysics 5.3

Thin-Film BAW Composite Resonator

Introduction

Bulk acoustic wave (BAW) resonators can be used as narrow band filters in radio-frequency applications. Their chief advantage compared with traditional ceramic electromagnetic resonators is that they can be made smaller in size because they can be designed to have an acoustic wavelength smaller than the electromagnetic wavelength.

In addition to the desired bulk acoustic mode, the resonator structure may have many spurious modes with very narrow spacing. The design goal is usually to maximize the quality of the main component and to reduce the effect of spurious modes.

This tutorial shows how you can model thin-film BAW resonators in 2D using eigenfrequency and frequency-response analyses. The geometry used here is the same as that in Ref. 1 and Ref. 2.

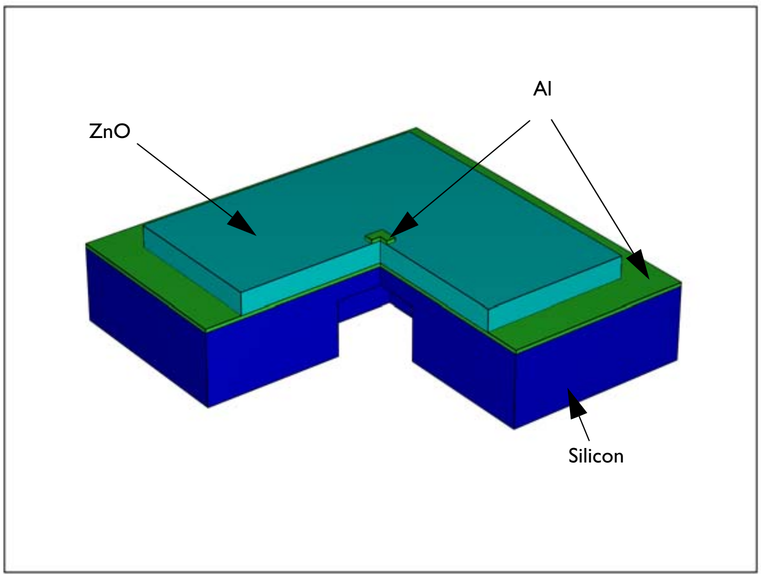


Figure 1: Arbitrarily scaled geometry of a thin film BAW resonator.

Model Definition

Figure 1 shows the geometry of the resonator from Ref. 1. The lowest layer of the resonator is silicon. On top of that, there is an aluminum layer that operates as the ground electrode. Above the aluminum layer is the active piezoelectric layer made of zinc oxide

(ZnO). The topmost layer of the resonator is an aluminum electrode. The material properties used in this model are obtained from the MEMS Module material library.

A large part of the silicon layer is etched away from the lower end of the central region of the resonator structure. This effectively reduces the thickness of the active central region thereby making the device a thin-film composite BAW resonator.

The thickness of the silicon layer at the central region is 7 μm. Both aluminum layers are 0.2 μm thick, and the piezoelectric layer is 9.5 μm thick. The width of the rectangular top electrode is 500 μm. The thin silicon area is roughly 1.7 mm wide.

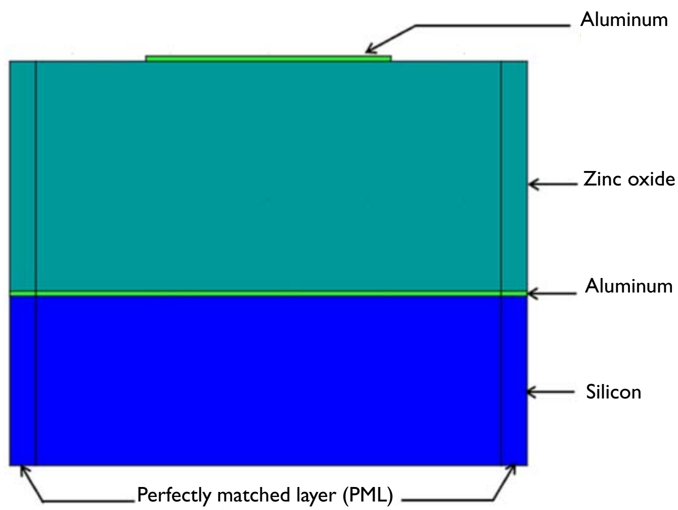


Figure 2: The 2D geometry (not drawn to scale) used in the tutorial.

This example is modeled in 2D, using the plane strain assumption where the out-of-plane thickness is specified to be 1.7 mm. The modeled geometry (Figure 2) is a symmetric 1-mm section in the center of the resonator. The Perfectly Matched Layer (PML) domains used on the two sides effectively increase the length of the resonator and simulates the effect of propagation and absorption of elastic waves in the adjoining regions which are not resolved in the true geometric scale.

The absorption of elastic waves in the PML domains contribute to the damping of the structure. This is also known as *anchor loss*. Additionally, the model also incorporates mechanical and electrical losses in the piezoelectric zinc oxide layer by means of loss factors. A structural loss factor represents the hysteresis in a stress-strain curve and a dielectric loss factor represents the polarization loss, which manifests itself as the hysteresis in the polarization versus electric field curve of the material. The structural and dielectric loss factors appear as the imaginary components of the mechanical stiffness and relative permittivity, respectively.

Ref. 3 gives the material quality Q_m and the dielectric loss tangent $\tan\delta$ for many materials. The magnitude of Q_m is roughly 100–1000, and the magnitude of $\tan\delta$ is roughly 0.001–0.01. Based on that data, the following values are used:

- Structural loss factor: $\eta_{cE} = 0.001$.
- Dielectric loss factor: $\eta_{\varepsilon S} = 0.01$.

In this model, COMSOL Multiphysics solves for both structural and electrical equations in the piezoelectric layer but only solves for the structural equation in the other layers. The electrical equations are not solved in the metallic aluminum layers because the electrical conductivity of aluminum is several orders of magnitude higher than that of zinc oxide and hence the aluminum layers almost act as equipotential regions allowing extremely small conduction current through them. Therefore the electrical characteristics of aluminum do not have any significant effect on the response of the resonator. The dominant electromechanical coupling is exhibited by the piezoelectric layer only.

This tutorial shows two different analyses. In the first step, you compute and investigate the eigenmodes of the structure, with its lateral ends fixed. In the second step, you analyze the frequency response of the resonator within the desired bandwidth of 215 MHz to 235 MHz.

Results and Discussion

Figure 3 shows the lowest BAW mode of the structure which occurs at 221.4 MHz. This plot was generated from the results of the eigenfrequency analysis. This is the fundamental longitudinal thickness mode. The plot shows scaled deformation only to be used for visualization of the mode shape. Note that COMSOL Multiphysics computes complex-valued eigenfrequencies where the imaginary part gives a measure of the damping due to structural loss, polarization loss and anchor loss.

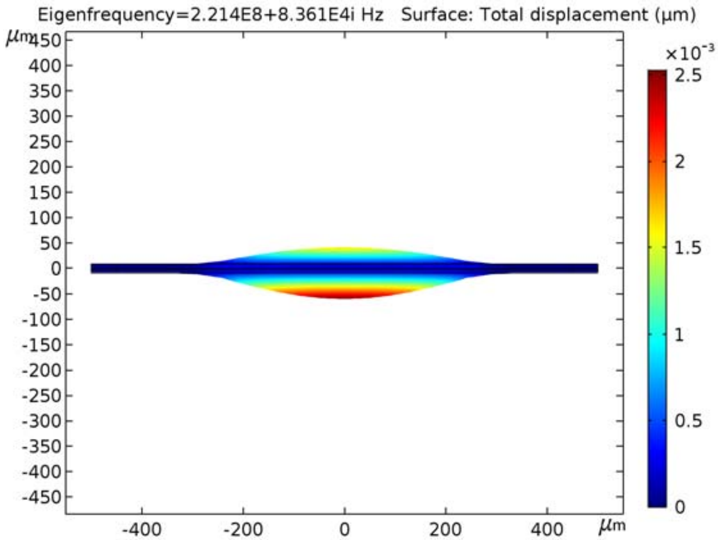


Figure 3: The lowest bulk acoustic mode of the resonator identified from the solutions of the eigenfrequency analysis.

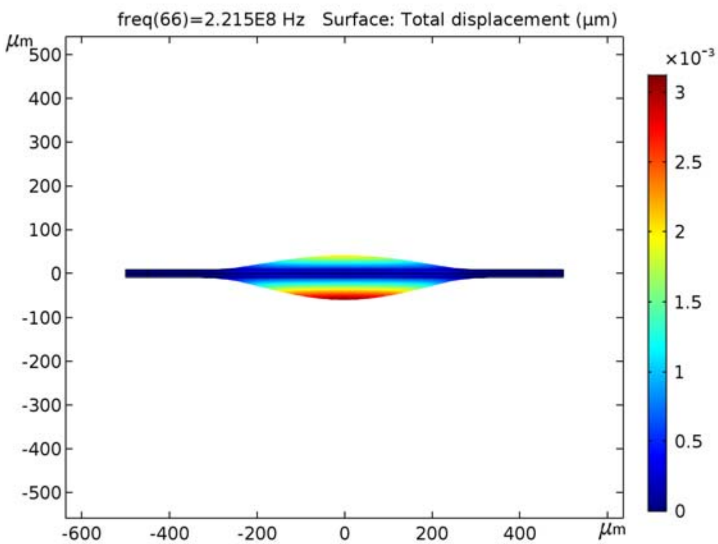


Figure 4: The lowest bulk acoustic mode of the resonator identified from the solutions of the frequency domain analysis.

Figure 4 shows the deformation of the resonator obtained from the frequency response analysis when the zinc oxide layer is excited with 1 volt (zero-to-peak voltage) at 221.5 MHz. The maximum deflection is about 3 nm.

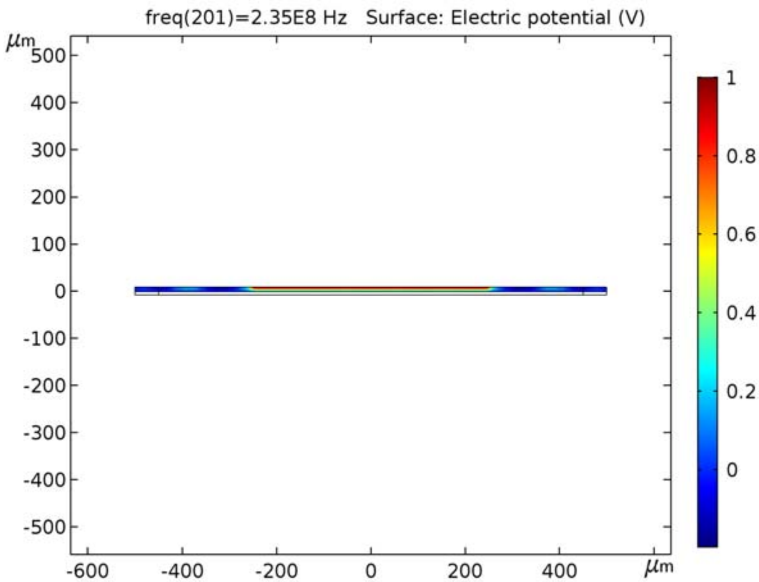


Figure 5: Electric potential distribution in the zinc oxide layer at 235 MHz excitation.

Figure 5 shows the voltage distribution in the piezoelectric layer when excited at 235 MHz.

COMSOL Multiphysics' Terminal boundary condition which is used to specify the voltage on the piezoelectric material also automatically computes the admittance. The admittance is the ratio of the total current flowing through the piezoelectric material to the voltage across it. It is a complex-valued quantity for a lossy material. Typically the imaginary part reflects the displacement current and the real part reflects the conduction current as well as other losses in the structure. Figure 6 shows the absolute value of admittance as a function of frequency. Within the investigated range of 215 MHz to 235 MHz, the admittance is very similar to that shown in Ref. 2. Note that the highest peak in admittance occurs at the lowest BAW mode of 221 MHz.

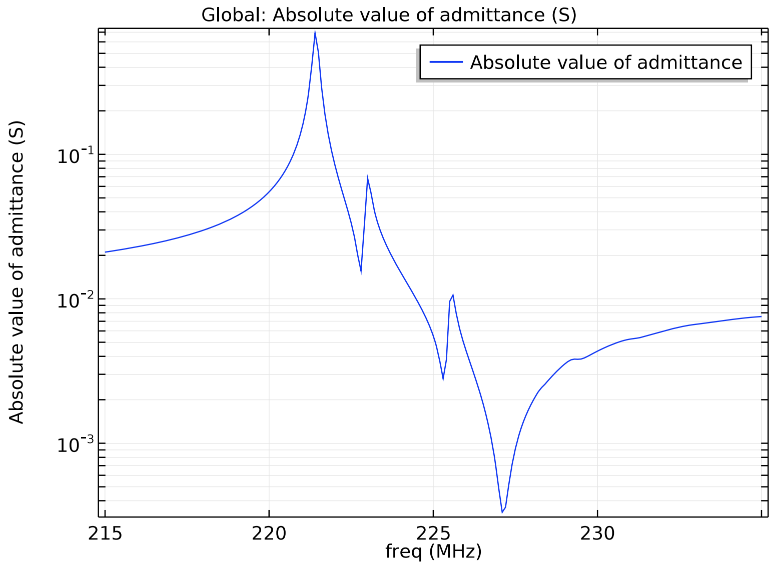


Figure 6: Absolute value of the admittance vs. frequency.

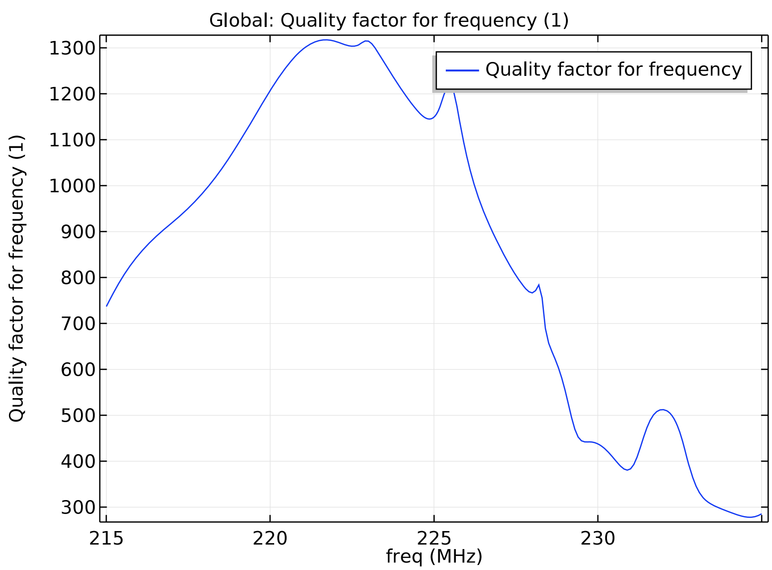


Figure 7: Quality factor vs. frequency.

Figure 7 shows the quality factor of the device as a function of frequency. The quality factor or Q-factor indicates the number of cycles (at the given frequency) in which the total energy of the system decreases by a factor of $e^{2\pi}$. This is also automatically computed by COMSOL Multiphysics. Figure 7 shows that the maximum value of Q ~ 1300 is obtained at around 221 MHz. This value obtained from the frequency-response analysis agrees well with the Q-factor computed by the eigenfrequency analysis. The results from the eigenfrequency analysis shows the Q-factor at 221.4 MHz to be 1326.

The eigenfrequency analysis also automatically computes the decay factor for each eigenfrequency. The inverse of the decay factor is the time required for the amplitude of a damped signal to reduce to e^{-1} of its initial amplitude. The decay factor at 221.4 MHz was computed to be $5.25 \cdot 10^5\ \mathrm{s}^{-1}$.

References

1. R.F. Milsom, J.E., Curran, S.L. Murray, S. Terry-Wood, and M. Redwood, "Effect of Mesa-Shaping on Spurious Modes in ZnO/Si Bulk-Wave Composite Resonators," *Proc. IEEE Ultrason. Symp.*, pp. 498–503, 1983.

2. T. Makkonen, A. Holappa, J. Ellä, and M.M. Salomaa, "Finite element simulations of thin-film composite BAW resonators," *IEEE Transactions on Ultrasonics, Ferroelectrics, and Frequency Control*, vol. 48, no. 5, 2001.

3. Morgan Advanced Materials, http://www.morganelectroceramics.com

Application Library path: `MEMS_Module/Piezoelectric_Devices/thin_film_baw_resonator`

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 **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Piezoelectric Devices**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Eigenfrequency**.

6 Click **Done**.

GEOMETRY 1

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**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `1000`.

4 In the **Height** text field, type `16.7`.

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

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

Layer name	Thickness (µm)
Layer 1	50

7 Select the **Layers to the left** check box.

8 Select the **Layers to the right** check box.

9 Clear the **Layers on bottom** check box.

10 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Rectangle 2 (r2)

1 Right-click **Rectangle 1 (r1)** and choose **Duplicate**.

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

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

4 Locate the **Position** section. In the **y** text field, type `-1.25`.

5 Right-click **Component 1 (comp1)>Geometry 1>Rectangle 2 (r2)** and choose **Build Selected**.

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `500`.

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

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

6 In the **y** text field, type `8.45`.

7 Click **Build All Objects**.

DEFINITIONS

In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

Axis

Change the aspect ratio to have a better view of the model, and make the domain selections easier.

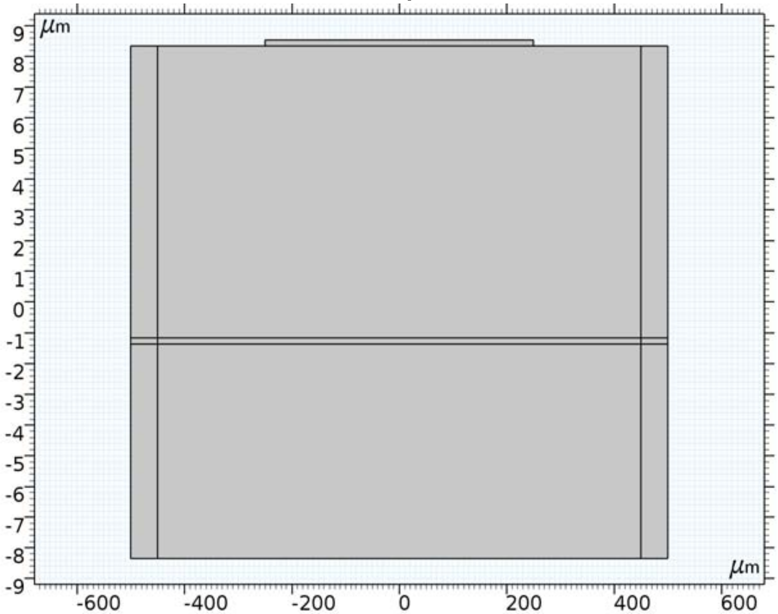
1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions>View 1** node, then click **Axis**.

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

3 From the **View scale** list, choose **Automatic**.

4 Click **Update**.

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



Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1-3,8-10` in the **Selection** text field.

5 Click **OK**.

In this way you specify that the layers on the two sides of the geometry form a perfectly matched layer.

ADD MATERIAL

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

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

3 In the tree, select **MEMS>Semiconductors>Si - Silicon (single-crystal, anisotropic)**.

4 Click **Add to Component** in the window toolbar.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **MEMS>Metals>Al - Aluminum / Aluminium**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Al - Aluminum / Aluminium (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Al - Aluminum / Aluminium (mat2)**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `2 5 7 9` in the **Selection** text field.

5 Click **OK**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Piezoelectric>Zinc Oxide**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Zinc Oxide (mat3)

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

2 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Zinc Oxide (mat3)**.

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `3 6 10` in the **Selection** text field.

6 Click **OK**.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Thickness** section.

3 In the d text field, type `1.7[mm]`.

Piezoelectric Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Piezoelectric Material 1**.

2 In the **Settings** window for **Piezoelectric Material**, locate the **Domain Selection** section.
3 Click **Clear Selection**.
4 Click **Paste Selection**.
5 In the **Paste Selection** dialog box, type `3 6 10` in the **Selection** text field.
6 Click **OK**.

These are piezoelectric domains.

Mechanical Damping 1

1 In the **Model Builder** window, right-click **Piezoelectric Material 1** and choose **Mechanical Damping**.
2 In the **Settings** window for **Mechanical Damping**, locate the **Damping Settings** section.
3 From the **Damping type** list, choose **Isotropic loss factor**.
4 From the η_s list, choose **User defined**. In the associated text field, type `0.001`.

Dielectric Loss 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** right-click **Piezoelectric Material 1** and choose **Dielectric Loss**.
2 In the **Settings** window for **Dielectric Loss**, locate the **Dielectric Loss Settings** section.
3 From the $\eta_{\varepsilon S}$ list, choose **User defined**. In the associated text field, type `0.01`.

Linear Elastic Material 2

1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Material Models>Linear Elastic Material**.
2 In the **Settings** window for **Linear Elastic Material**, locate the **Domain Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type `1 4 8` in the **Selection** text field.
5 Click **OK**.
6 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.
7 From the **Solid model** list, choose **Anisotropic**.

These are the silicon domains that are modeled as an anisotropic linear elastic material.

The remaining domains are the aluminum domains that are modeled as an isotropic linear elastic material.

Fixed Constraint 1

1 Right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1 3 5 27-29` in the **Selection** text field.

5 Click **OK**.

These are the boundaries of the perfectly matched layers. In this way you indicate that the device is fixed on its sides that are far away from the region that you are modeling.

ELECTROSTATICS (ES)

On the **Physics** toolbar, click **Solid Mechanics (solid)** and choose **Electrostatics (es)**.

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

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `3 6 10` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Electrostatics**, locate the **Thickness** section.

8 In the d text field, type `1.7[mm]`.

Ground 1

1 Right-click **Component 1 (comp1)>Electrostatics (es)** and choose **Ground**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `6 13 25` in the **Selection** text field.

5 Click **OK**.

Terminal 1

1 In the **Model Builder** window, right-click **Electrostatics (es)** and choose the boundary condition **Terminal**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `16` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Terminal**, locate the **Terminal** section.

7 From the **Terminal type** list, choose **Voltage**.

MESH 1

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `2 21` in the **Selection** text field.

6 Click **OK**.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `9` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Distribution**, locate the **Distribution** section.

7 In the **Number of elements** text field, type `100`.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `8` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Distribution**, locate the **Distribution** section.

7 In the **Number of elements** text field, type `2`.

Distribution 4

1 Right-click **Mapped 1** and choose **Distribution**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `12` in the **Selection** text field.

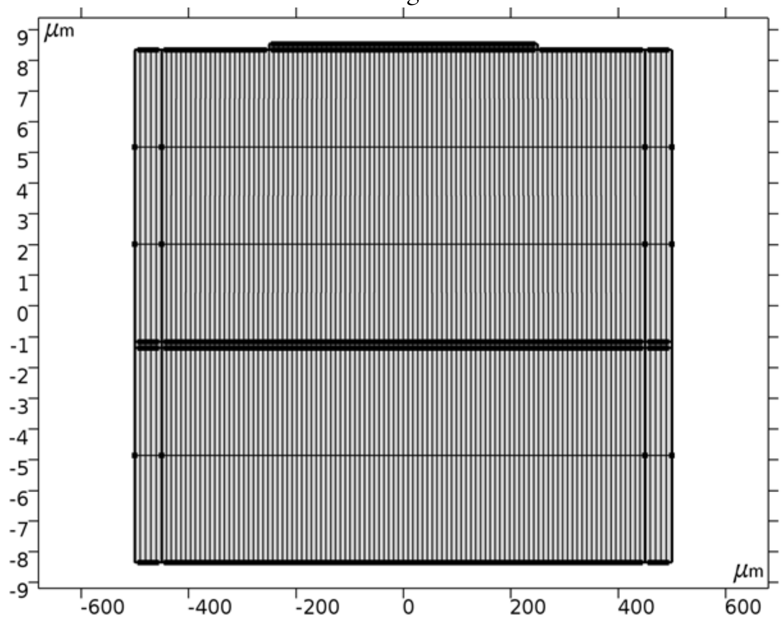
5 Click **OK**.

6 In the **Settings** window for **Distribution**, locate the **Distribution** section.

7 In the **Number of elements** text field, type `3`.

8 Click **Build All**.

The mesh should look as shown in this figure.



STUDY 1

Step 1: Eigenfrequency

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

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

3 Select the **Search for eigenfrequencies around** check box.

4 In the associated text field, type `220[MHz]+i*150[kHz]`.

5 From the **Eigenfrequency search method around shift** list, choose **Smaller imaginary part**.

6 Right-click **Study 1>Step 1: Eigenfrequency** and choose **Get Initial Value for Step**.

7 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)** node, then click **Eigenvalue Solver 1**.

2 In the **Settings** window for **Eigenvalue Solver**, locate the **Values of Linearization Point** section.

3 Find the **Value of eigenvalue linearization point** subsection. In the **Point** text field, type `2e8`.

4 Click **Compute**.

DEFINITIONS

Axis

Set the view to the default scale type for the result plots.

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>View 1** click **Axis**.

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

3 From the **View scale** list, choose **None**.

4 Click **Update**.

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

RESULTS

Mode Shape (solid)

1 In the **Model Builder** window, under **Results** click **Mode Shape (solid)**.

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

3 From the **Eigenfrequency (Hz)** list, choose **2.214E8+8.361E4i**.

4 On the **Mode Shape (solid)** toolbar, click **Plot**.

Compare this plot with .

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> Frequency Domain**.

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

STUDY 2

Step 1: Frequency Domain

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

2 In the **Model Builder** window, click **Step 1: Frequency Domain**.

3 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

4 In the **Frequencies** text field, type `range(215,0.1,235)[MHz]`.

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

RESULTS

Stress (solid)

1 In the **Model Builder** window, click **Stress (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Displacement (solid)` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **2.215E8**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Displacement (solid)** node, then click **Surface 1**.

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>Solid Mechanics> Displacement>solid.disp - Total displacement**.

3 On the **Displacement (solid)** toolbar, click **Plot**.

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

Compare this plot with .

Electric Potential (es) 1

1 In the **Model Builder** window, under **Results** click **Electric Potential (es) 1**.

2 On the **Electric Potential (es) 1** toolbar, click **Plot**.

Compare this plot with .

1D Plot Group 5

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

2 In the **Settings** window for **1D Plot Group**, type `Admittance` in the **Label** text field.

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

Global 1

1 Right-click **Admittance** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
abs(es.Y11)	S	Absolute value of admittance

4 Locate the **x-Axis Data** section. From the **Unit** list, choose **MHz**.

5 On the **Admittance** toolbar, click **Plot**.

6 Click the **y-Axis Log Scale** button on the **Graphics** toolbar.

Compare this plot with .

Admittance 1

1 In the **Model Builder** window, under **Results** right-click **Admittance** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Quality Factor` in the **Label** text field.

Global 1

1 In the **Model Builder** window, expand the **Results>Quality Factor** node, then click **Global 1**.

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>Solid Mechanics>Global> solid.Q_freq - Quality factor for frequency**.

3 Click the **y-Axis Log Scale** button on the **Graphics** toolbar.

4 On the **Quality Factor** toolbar, click **Plot**.

Compare this plot with .

The following steps show how to compute the quality factor and the decay factor for the resonance at 221.4 MHz from the eigenfrequency study. You will see that the value for the quality factor agrees well with that obtained from the frequency domain study at the same frequency.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Q-Factor` in the **Label** text field.

3 Locate the **Data** section. From the **Eigenfrequency selection** list, choose **From list**.

4 In the **Eigenfrequency (Hz)** list, select **2.214E8+8.361E4i**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Solid Mechanics>Global>solid.Q_eig - Quality factor for eigenvalue**.

6 Click **Evaluate**.

Q-Factor 1

1 Right-click **Q-Factor** and choose **Duplicate**.

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>Solid Mechanics>Global>solid.decay - Exponential decay factor**.

3 In the **Label** text field, type `Decay Factor`.

4 Click **Evaluate**.