Created in COMSOL Multiphysics 5.6

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. The capability of the Electromechanics multiphysics interface to handle both conducting and dielectric materials is also demonstrated.

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 (dielectric constant 7.5). Beneath the substrate is a silicon counter electrode that is grounded. The suspended plate is structurally anchored to the substrate by four rectangular flexures at its corners but is electrically isolated from the substrate.

Initially a small potential of 1 mV is applied to the polysilicon using the Domain Terminal feature. 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 gap between the polysilicon and the nitride presents a model setup challenge. Since it is not possible to collapse the mesh to zero thickness (when the gap is closed) for numerical reasons, 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 domain that contains both the nitride and the gap. Specifically, the dielectric constant in the domain is represented by a smoothed step function. The midpoint of the smoothed step function 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 domain takes the value of the nitride dielectric constant throughout the domain.

It is also worth noting that the Electromechanics multiphysics interface is capable of treating both conductors (for example, the polysilicon) and dielectric materials (for example, the nitride), as demonstrated in this model.

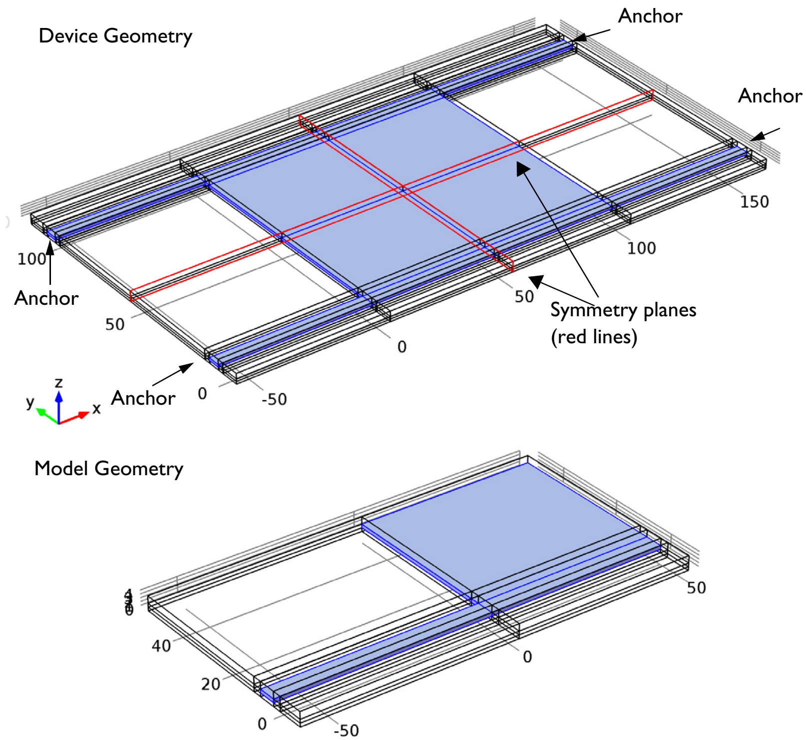

Figure 1: Top: Device geometry showing anchor points and symmetry planes. Bottom: Model geometry. Due to symmetry only one quadrant of the device needs to be modeled.

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:

$$F_c = t_n - e_n \cdot g \qquad g < 0$$

$$F_c = t_n + \exp\left(-\frac{e_n}{t_n} \cdot g\right) \qquad g \geq 0$$

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 tune the elastic stiffness and the contact force. The formula for the contact force is an approximate one and the model 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.

For this same reason, a small mechanical damping is added to the Si domain to suppress the ringing of the structure once contact is made, thus preventing the time dependent solver from taking very small steps to resolve the ringing, which is not the focus of this model. (Of course care must be taken to ensure that the magnitude of the extra damping should not change the switching time constant significantly.)

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.

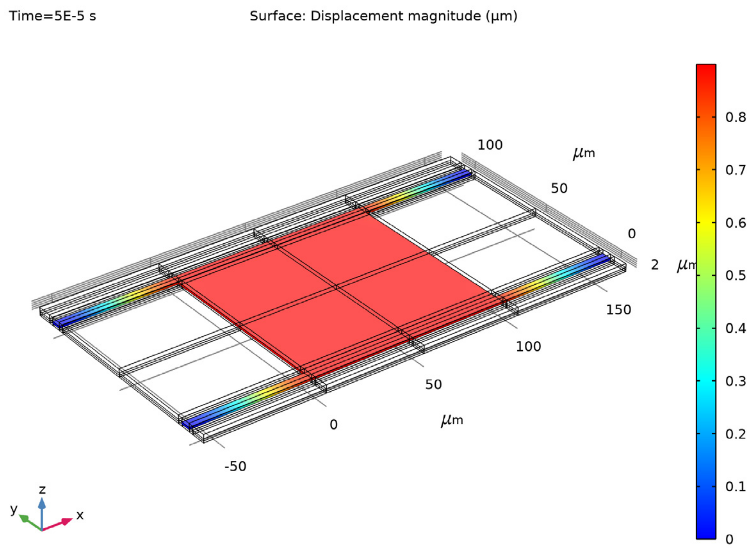

Figure 2: Displacement of the polysilicon when pulled in. Most of the polysilicon structure is in contact with the silicon nitride with a displacement of 0.9 µm.

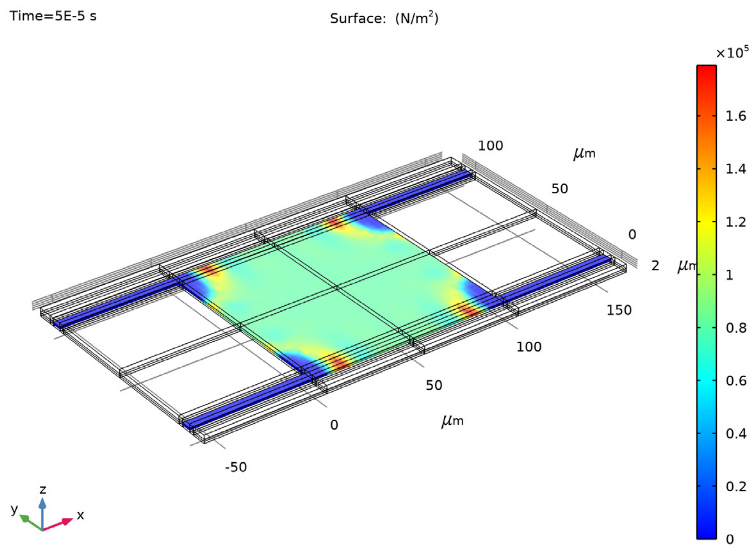

Figure 3: Contact forces acting on the polysilicon when the structure is pulled in.

Figure 4 shows the displacement of the switch as a function of time. The switch takes significantly longer to close than the time scale on which the applied 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 65. Note that the capacitance changes on a significantly shorter time scale than the displacement.

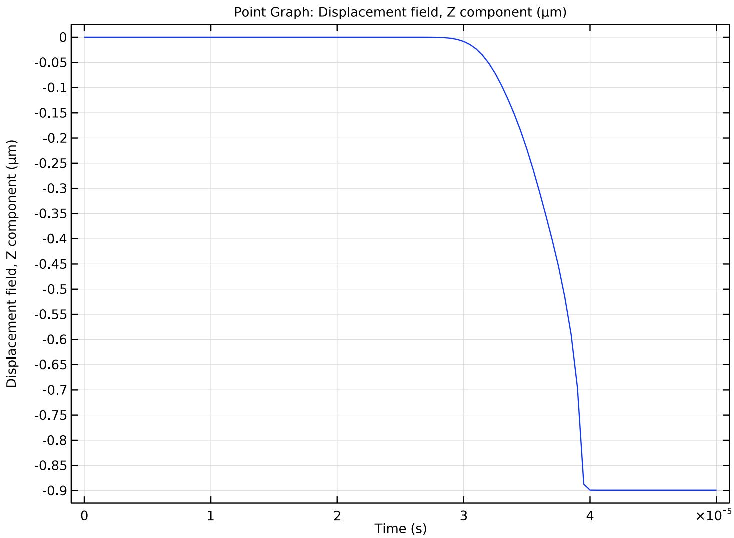

Figure 4: Displacement of the center of the device as a function of time.

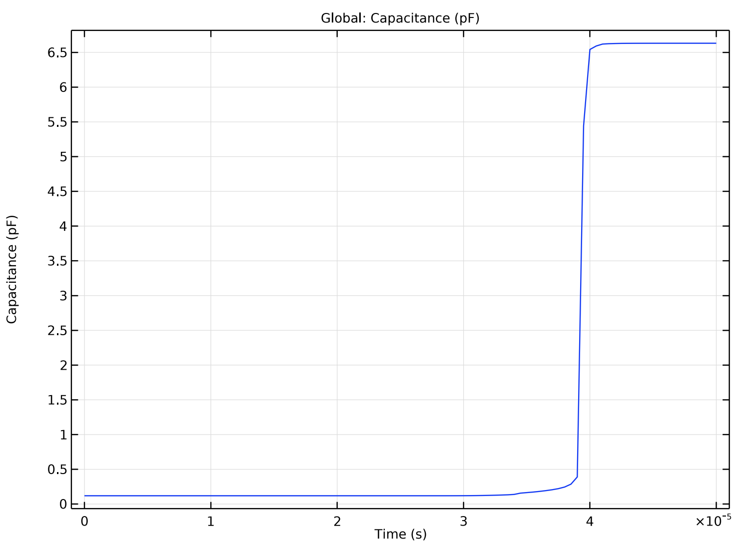

Figure 5: Capacitance of the device as a function of time. The transients in this plot that occur after the point of contact are not physical. The capacitance of the structure changes from 0.1 pF to 6.5 pF as a result of the pull-in.

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>Electromagnetics-Structure Interaction>Electromechanics>Electromechanics**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General 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

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
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 In the **Home** toolbar, click a= **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])		

GEOMETRY 1

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

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

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

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

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

3 In the **Width** text field, type 110.

4 In the **Height** text field, type 5.

5 Locate the **Position** section. In the **xw** text field, type -60.

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

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

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

3 In the **Width** text field, type `50`.

4 In the **Height** text field, type `60`.

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

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

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

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

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

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

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

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

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

Extrude 1 (ext1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **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 Click **Build Selected**.

Form Union (fin)

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

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

DEFINITIONS

Step 1 (step1)

1 In the **Home** toolbar, click f(X) **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 In the **Home** toolbar, click f(X) **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 Locate the **Smoothing** section. In the **Size of transition zone** text field, type `1e-5`.

Define selections.

Bridge

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

2 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.

Gap

1 In 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**.

Bridge surface

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

2 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**.

Base

1 In 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 In 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**.

Bridge lower side

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

2 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**.

Symmetry x

1 In 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**.

Symmetry y

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

2 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**.

Symmetry

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

2 In the **Settings** window for **Union**, 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, in the **Selections to add** list, choose **Symmetry x** and **Symmetry y**.

6 Click **OK**.

7 In the **Settings** window for **Union**, type `Symmetry` in the **Label** text field.

All domains

1 In 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 In the **Label** text field, type `All domains`.

Non-solid

1 In 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 **Bridge** in the **Selections to subtract** list.

9 Click **OK**.

10 In the **Settings** window for **Difference**, type `Non-solid` in the **Label** text field.

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 **Domain Selection** section.

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

Symmetry 1

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

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

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

Boundary Load 1

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

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

3 From the **Selection** list, choose **Bridge lower side**.

4 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

contactpressure	z

Fixed Constraint 1

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

2 Select Boundary 24 only.

Linear Elastic Material 1

Add some small damping to reduce the transient effects at the final stage of the switch motion.

1 In the **Model Builder** window, click **Linear Elastic Material 1**.

Damping 1

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

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

3 In the β_{dK} text field, type `1e-6`.

4 Locate the **Domain Selection** section. From the **Selection** list, choose **Bridge**.

ELECTROSTATICS (ES)

The default **Charge Conservation**feature was set to use solid material type. Add one more feature to represent the nonsolid (air) domains.

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

Charge Conservation, Non-solid

1 In the **Physics** toolbar, click **Domains** and choose **Charge Conservation**.

2 In the **Settings** window for **Charge Conservation**, type `Charge Conservation, Non-solid` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Non-solid**.

Define the voltage applied to the bridge.

Terminal 1

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

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

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

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

5 In the V_0 text field, type `Va`.

Define the voltage applied to the base.

Terminal 2

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

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

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

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

5 In the V_0 text field, type `0`.

Terminal 1

Add another **Domain Terminal** to the bridge. 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 **Model Builder** window, under **Component 1 (comp1)>Electrostatics (es)** right-click **Terminal 1** and choose **Duplicate**.

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

3 In the V_0 text field, type `V0`.

DEFINITIONS

Deforming Domain 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Moving Mesh** click **Deforming Domain 1**.

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

3 From the **Selection** list, choose **Non-solid**.

Symmetry/Roller 1

1 In the **Model Builder** window, click **Symmetry/Roller 1**.

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

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

Load the materials.

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Air**.

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

5 In the tree, select **MEMS>Semiconductors>Si - Polycrystalline silicon**.

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

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

MATERIALS

Si - Polycrystalline silicon (mat2)

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

2 From the **Selection** list, choose **Bridge**.

Material 3 (mat3)

1 In the **Model Builder** window, 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. From the **Material type** list, choose **Nonsolid**.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	`1+6.5*(1-step1(z/1[m]))`	1	Basic

MESH 1

Mapped 1

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

2 Select Boundaries 10, 20, 30, 40, 50, 63, 73, 83, 93, and 103 only.

Swept 1

In the **Mesh** toolbar, click **Swept**.

Distribution 1

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

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

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

4 Locate the **Distribution** section. In the **Number of elements** text field, type `4`.

Distribution 2

1 In the **Model Builder** window, right-click **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 `2`.

4 Select Domains 2, 5, 8, 11, 14, 23, 26, and 29 only.

5 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

Time Dependent

1 In 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 **Output times** text field, type `range(0,5e-7,5e-5)`.

4 Click to expand the **Results While Solving** section. Select the **Plot** check box.

5 From the **Update at** list, choose **Time steps taken by solver**.

6 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

7 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Electrostatics (es)>Terminal 3**.

8 Click **Disable**.

Solution 1 (sol1)

1 In 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 From the **Steps taken by solver** list, choose **Intermediate**.

5 Right-click **Time-Dependent Solver 1** and choose **Fully Coupled**.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Time-Dependent Solver 1** click **Fully Coupled 1**.

7 In the **Settings** window for **Fully Coupled**, click **Compute**.

RESULTS

Study 1/Solution 1 (sol1)

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

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

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

Mirror the solution to visualize the whole structure.

Mirror 3D 1

1 In the **Results** toolbar, click **More Datasets** 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 **Dataset** 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 (solid)

1 In the **Model Builder** window, click **Displacement (solid)**.

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

3 From the **Dataset** list, choose **Mirror 3D 2**.

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

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

Electric Potential (es)

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

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

3 From the **Dataset** list, choose **Mirror 3D 2**.

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

7 In the **Electric Potential (es)** toolbar, click **Plot**.

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

Contact force

1 In the **Model Builder** window, right-click **Displacement (solid)** and choose **Duplicate**.

2 Right-click **Displacement (solid) 1** and choose **Rename**.

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

4 Click **OK**.

Surface 1

1 In the **Model Builder** window, expand the **Contact force** 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 In the **Contact force** toolbar, click **Plot**.

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

Displacement

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

2 Right-click **1D Plot Group 5** 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 In the **Displacement** toolbar, click **Plot**.

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

Capacitance

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

2 Right-click **1D Plot Group 6** 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*es.Q0_1/es.V0_1`	`pF`	`Capacitance`

4 Click to expand the **Legends** section. Clear the **Show legends** check box.

5 In the **Capacitance** toolbar, click **Plot**.

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