

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 $0.9\ \mu\text{m}$ above a $0.1\ \mu\text{m}$ 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 $25\ \mu\text{s}$ the applied voltage is increased by 5 V with a step function that has a rise time of $10\ \mu\text{s}$. 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.

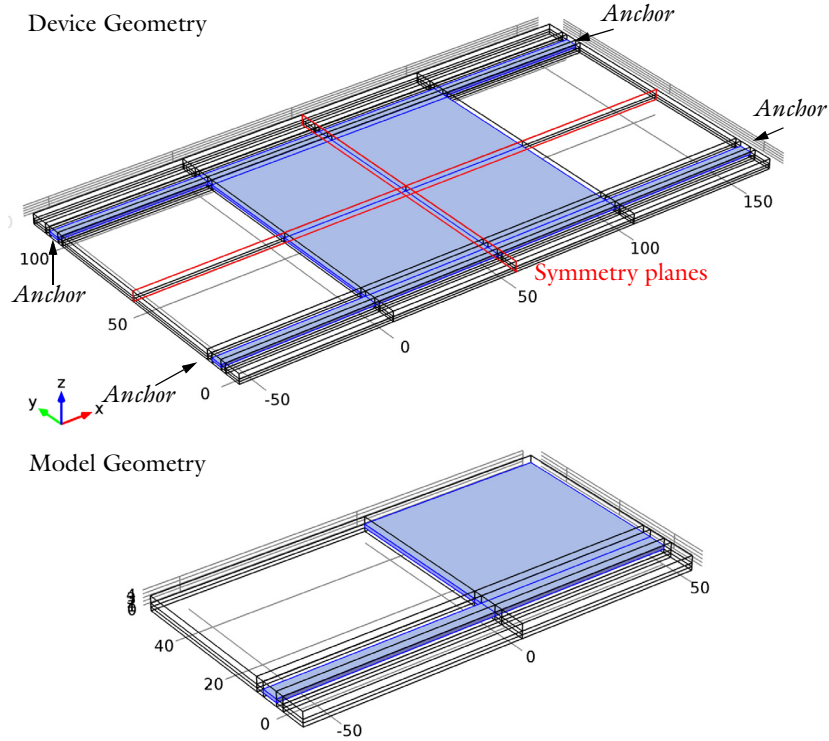


Figure 1: Top: Device geometry showing anchor points and symmetry planes. Bottom: Model geometry. Due to symmetry only one quadrant of the device needs to be modeled.

The contact between the polysilicon and the nitride is handled by an approximate penalty or barrier method, as described in Ref. 1. Stiff, non-linear 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 F_c are given by:

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

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

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.

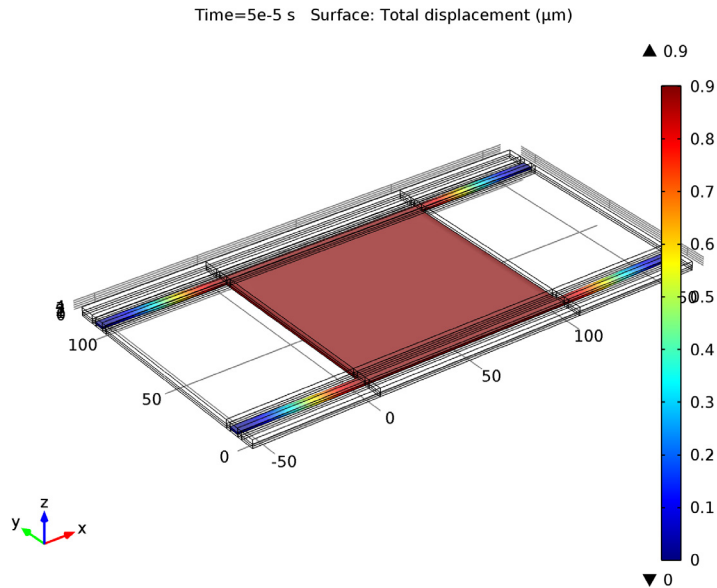


Figure 2: Displacement of the polysilicon when pulled in. Most of the polysilicon structure is in contact with the silicon nitride with a displacement of $0.9 \mu\text{m}$.

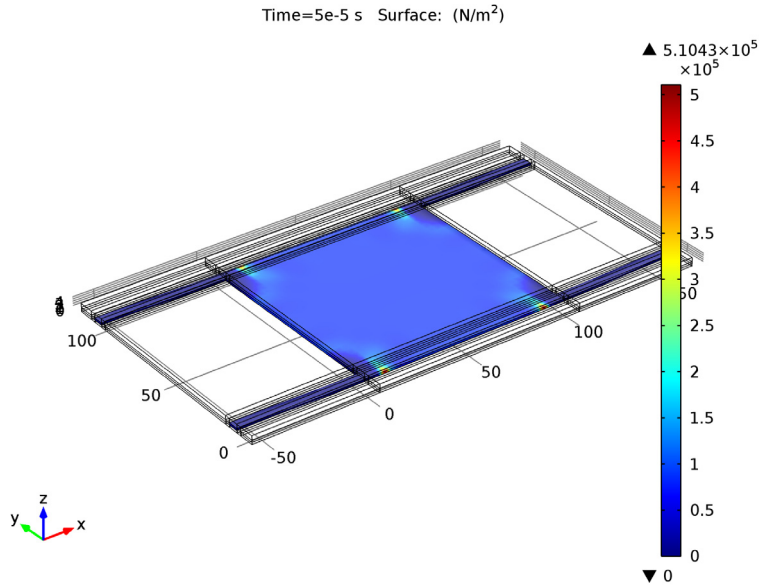


Figure 3: Contact forces acting on the polysilicon when the structure is pulled in.

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.

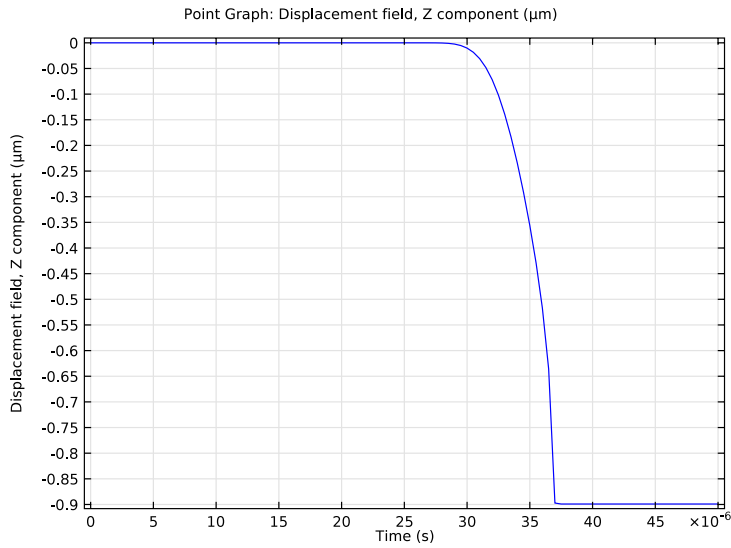


Figure 4: Displacement of the center of the device as a function of time.

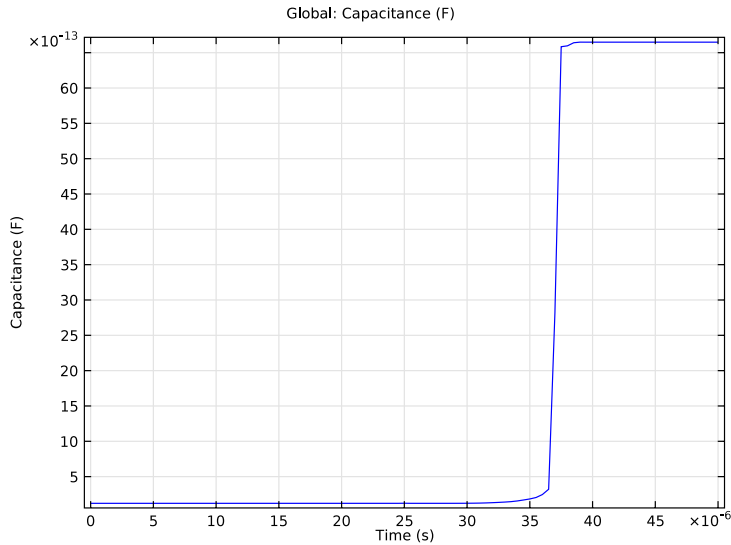


Figure 5: Capacitance of the device as a function of time. The transients in this plot that occur after the point of contact are not physical. The capacitance of the structure changes from 0.1 pF to 1.5 pF as a result of the pull-in.

Reference

1. Crisfield M. A., *Non-linear Finite Element Analysis of Solids and Structures, volume 2: Advanced Topics*, John Wiley & Sons Ltd., England, 1997.
-

Model Library path: MEMS_Module/Actuators/rf_mems_switch

Modeling Instructions

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

NEW

- 1 In the **New** window, click the **Model Wizard** button.

MODEL WIZARD

- 1 In the **Model Wizard** window, click the **3D** button.
- 2 In the **Select physics** tree, select **Structural Mechanics>Electromechanics (emi)**.
- 3 Click the **Add** button.
- 4 Click the **Study** button.
- 5 In the tree, select **Preset Studies>Stationary**.
- 6 Click the **Done** button.

GEOMETRY I

- 1 In the **Model Builder** window, under **Component 1** click **Geometry 1**.
- 2 In the **Geometry** settings window, 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 **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
V0	1[mV]	0.0010000 V	Initial voltage
Vstep	5[V]	5.0000 V	Voltage step
insheight	100[nm]	1.0000E-7 m	Insulator height
airheight	900[nm]	9.0000E-7 m	Air height
en	1e15[Pa/m]	1.0000E15 N/m ³	Spring stiffness
tn	5e5[Pa]	5.0000E5 Pa	Contact force

DEFINITIONS

Variables 1

- 1 In the **Model Builder** window, under **Component 1** right-click **Definitions** and choose **Variables**.
- 2 In the **Variables** settings window, locate the **Variables** section.
- 3 In the table, enter the following settings:

Name	Expression	Unit	Description
gap	airheight+w	m	
contactpressure	$(\text{gap} \leq 0) * (\text{tn} - \text{en} * \text{gap}) + (\text{gap} > 0) * \text{tn} * \exp(-\text{gap} * \text{en} / \text{tn})$	N/m ²	
Va	$V0 + V\text{step} * \text{step2}(t/1[\text{s}])$	V	

GEOMETRY 1

Plane Geometry

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

Rectangle 1

- 1 In the **Model Builder** window, under **Component 1 > Geometry 1 > Work Plane 1** right-click **Plane Geometry** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Width** edit field, type 110.
- 4 In the **Height** edit field, type 5.
- 5 Locate the **Position** section. In the **xw** edit field, type -60.

Rectangle 2

- 1 Right-click **Plane Geometry** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Width** edit field, type 50.
- 4 In the **Height** edit field, type 60.
- 5 Locate the **Position** section. In the **yw** edit field, type -10.

Rectangle 3

- 1 Right-click **Plane Geometry** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Width** edit field, type 110.
- 4 In the **Height** edit field, type 60.
- 5 Locate the **Position** section. In the **xw** edit field, type -60.
- 6 In the **yw** edit field, type -10.

Rectangle 4

- 1 Right-click **Plane Geometry** and choose **Rectangle**.
- 2 In the **Rectangle** settings window, locate the **Size** section.
- 3 In the **Width** edit field, type 110.
- 4 In the **Height** edit field, type 10.
- 5 Locate the **Position** section. In the **xw** edit field, type -60.
- 6 In the **yw** edit field, type -2.5.

Extrude 1

- 1 On the **Geometry** toolbar, click **Extrude**.
- 2 In the **Extrude** settings window, locate the **Distances from Plane** section.
- 3 In the table, enter the following settings:

Distances (μm)
2
4

- 4 Click the **Build Selected** button.

Form Union

- 1 On the **Home** toolbar, click **Build All**.
- 2 Click the **Wireframe Rendering** button on the Graphics toolbar.

DEFINITIONS

Step 1

- 1 On the **Home** toolbar, click **Functions** and choose **Global>Step**.
- 2 In the **Step** settings window, locate the **Parameters** section.
- 3 In the **Location** edit field, type $1.05 \cdot \text{insheight}$.
- 4 Click to expand the **Smoothing** section. In the **Size of transition zone** edit field, type $0.05 \cdot \text{insheight}$.

Step 2

- 1 On the **Home** toolbar, click **Functions** and choose **Global>Step**.
- 2 In the **Step** settings window, locate the **Parameters** section.
- 3 In the **Location** edit field, type $3e-5$.
- 4 Click to expand the **Smoothing** section. In the **Size of transition zone** edit field, type $1e-5$.

Define selections.

Explicit 1

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 In the **Model Builder** window, under **Component 1>Definitions** right-click **Explicit 1** and choose **Rename**.
- 3 Go to the **Rename Explicit** dialog box and type Bridge in the **New name** edit 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, under **Component 1>Definitions** right-click **Box 1** and choose **Rename**.
- 3 Go to the **Rename Box** dialog box and type Gap in the **New name** edit field.
- 4 Click **OK**.
- 5 In the **Box** settings window, locate the **Box Limits** section.
- 6 In the **z maximum** edit 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, under **Component 1>Definitions** right-click **Adjacent 1** and choose **Rename**.
- 3 Go to the **Rename Adjacent** dialog box and type Bridge surface in the **New name** edit field.
- 4 Click **OK**.
- 5 In the **Adjacent** settings window, locate the **Input Entities** section.
- 6 Under **Input selections**, click **Add**.
- 7 Go to the **Add** dialog box.
- 8 In the **Input selections** list, select **Bridge**.
- 9 Click the **OK** button.

Box 2

- 1 On the **Definitions** toolbar, click **Box**.
- 2 In the **Model Builder** window, under **Component 1>Definitions** right-click **Box 2** and choose **Rename**.
- 3 Go to the **Rename Box** dialog box and type Base in the **New name** edit field.
- 4 Click **OK**.
- 5 In the **Box** settings window, locate the **Geometric Entity Level** section.
- 6 From the **Level** list, choose **Boundary**.
- 7 Locate the **Box Limits** section. In the **z maximum** edit 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 **Box** settings window, locate the **Geometric Entity Level** section.
- 3 From the **Level** list, choose **Boundary**.
- 4 Locate the **Box Limits** section. In the **z minimum** edit field, type 0.9.
- 5 In the **z maximum** edit 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, under **Component 1 > Definitions** right-click **Intersection 1** and choose **Rename**.
- 3 Go to the **Rename Intersection** dialog box and type Bridge lower side in the **New name** edit field.
- 4 Click **OK**.
- 5 In the **Intersection** settings window, 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 Go to the **Add** dialog box.
- 9 In the **Selections to intersect** list, choose **Bridge surface** and **Box 3**.
- 10 Click the **OK** button.

Box 4

- 1 On the **Definitions** toolbar, click **Box**.
- 2 In the **Model Builder** window, under **Component 1 > Definitions** right-click **Box 4** and choose **Rename**.
- 3 Go to the **Rename Box** dialog box and type Symmetry x in the **New name** edit field.
- 4 Click **OK**.
- 5 In the **Box** settings window, locate the **Geometric Entity Level** section.
- 6 From the **Level** list, choose **Boundary**.
- 7 Locate the **Box Limits** section. In the **x minimum** edit 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, under **Component 1 > Definitions** right-click **Box 5** and choose **Rename**.
- 3 Go to the **Rename Box** dialog box and type Symmetry y in the **New name** edit field.
- 4 Click **OK**.
- 5 In the **Box** settings window, locate the **Geometric Entity Level** section.
- 6 From the **Level** list, choose **Boundary**.

- 7 Locate the **Box Limits** section. In the **y minimum** edit field, type 45.
- 8 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

MATERIALS

Load the materials.

On the **Home** toolbar, click **Add Material**.

ADD MATERIAL

- 1 Go to the **Add Material** window.
- 2 In the tree, select **Built-In>Air**.
- 3 In the **Add material** window, click **Add to Component**.

ADD MATERIAL

- 1 Go to the **Add Material** window.
- 2 In the tree, select **MEMS>Semiconductors>Poly-Si**.
- 3 In the **Add material** window, click **Add to Component**.

MATERIALS

Poly-Si (mat2)

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Poly-Si (mat2)**.
- 2 In the **Material** settings window, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Bridge**.

Material 3 (mat3)

- 1 In the **Model Builder** window, right-click **Materials** and choose **New Material**.
- 2 In the **Material** settings window, 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 **Non-solid**.
- 5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilon _r	7.5-step1(z)*6.5		Basic

ELECTROMECHANICS (EMI)*Linear Elastic Material 1*

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

Fixed Constraint 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.
- 2 Select Boundary 24 only.

Symmetry 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
- 2 In the **Symmetry** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Symmetry x**.

Symmetry 2

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
- 2 In the **Symmetry** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Symmetry y**.

Boundary Load 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
- 2 Select Boundaries 26, 79, 89, and 99 only.
- 3 In the **Boundary Load** settings window, locate the **Force** section.
- 4 Specify the \mathbf{F}_A vector as

0	x
0	y
contactpressure	z

Prescribed Mesh Displacement 2

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.
- 2 In the **Prescribed Mesh Displacement** settings window, 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 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.
- 2 In the **Prescribed Mesh Displacement** settings window, 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.

Terminal 1

Define the voltage applied to the bridge lower side.

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Terminal**.
- 2 In the **Terminal** settings window, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Bridge surface**.
- 4 Locate the **Terminal** section. From the **Terminal type** list, choose **Voltage**.
- 5 In the V_0 edit field, type V_a .

Terminal 2

Define the voltage applied to the bridge lower side.

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Terminal**.
- 2 In the **Terminal** settings window, 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 edit field, type 0.

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.

- 6 On the **Physics** toolbar, click **Boundaries** and choose **Terminal**.

Terminal 3

- 1 In the **Terminal** settings window, 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 edit field, type V_0 .

MESH 1

Mapped 1

- 1 In the **Model Builder** window, under **Component 1** right-click **Mesh 1** and choose **More Operations>Mapped**.
- 2 Select Boundaries 10, 20, 30, 40, 50, 63, 73, 83, 93, and 103 only.

Swept 1

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

Distribution 1

- 1 In the **Model Builder** window, under **Component 1**)>**Mesh 1** right-click **Swept 1** and choose **Distribution**.
- 2 In the **Distribution** settings window, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Gap**.
- 4 Locate the **Distribution** section. In the **Number of elements** edit field, type 8.
- 5 Click the **Build All** button.

STUDY 1

Define the study steps. The first step (stationary) is used to define the initial conditions of the transient problem (step 2).

Step 2: Time Dependent

- 1 On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.
- 2 In the **Time Dependent** settings window, locate the **Study Settings** section.
- 3 In the **Times** edit 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** > **Electromechanics>Terminal 3**.
- 6 Click **Disable**.

Change the default solver to a fully coupled direct solver.

Solver 1

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solver 1** node.

- 3 Right-click **Time-Dependent Solver 1** and choose **Fully Coupled**.
- 4 In the **Fully Coupled** settings window, click to expand the **Method and termination** section.
- 5 Locate the **Method and Termination** section. From the **Nonlinear method** list, choose **Automatic (Newton)**.
- 6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Time-Dependent Solver 1** right-click **Direct** and choose **Enable**.
- 7 In the **Direct** settings window, locate the **General** section.
- 8 From the **Solver** list, choose **PARDISO**.
- 9 Click **Study 1>Solver Configurations>Solver 1>Time-Dependent Solver 1>Direct** and click **Compute**.

RESULTS

Create the mirror solutions.

Data Sets

- 1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.
- 2 In the **Mirror 3D** settings window, locate the **Plane Data** section.
- 3 In the **x-coordinate** edit field, type 50.
- 4 Right-click **Results>Data Sets>Mirror 3D 1** and choose **Duplicate**.
- 5 In the **Mirror 3D** settings window, locate the **Data** section.
- 6 From the **Data set** list, choose **Mirror 3D 1**.
- 7 Locate the **Plane Data** section. From the **Plane** list, choose **xz-planes**.
- 8 In the **y-coordinate** edit field, type 50.

Displacement (emi)

- 1 In the **Model Builder** window, under **Results** click **Displacement (emi)**.
- 2 In the **3D Plot Group** settings window, locate the **Data** section.
- 3 From the **Data set** list, choose **Mirror 3D 2**.
- 4 On the **3D Plot Group** toolbar, click **Plot**.

Potential (emi)

- 1 In the **Model Builder** window, under **Results** click **Potential (emi)**.
- 2 In the **3D Plot Group** settings window, locate the **Data** section.
- 3 From the **Data set** list, choose **Mirror 3D 2**.

- 4 On the **3D Plot Group** toolbar, click **Plot**.

Displacement (emi) I

- 1 In the **Model Builder** window, under **Results** right-click **Displacement (emi)** and choose **Duplicate**.
- 2 Right-click **Displacement (emi) I** and choose **Rename**.
- 3 Go to the **Rename 3D Plot Group** dialog box and type **Contact force (emi)** in the **New name** edit field.
- 4 Click **OK**.

Contact force (emi)

- 1 In the **Model Builder** window, expand the **Results>Contact force (emi)** node, then click **Surface I**.
- 2 In the **Surface** settings window, locate the **Expression** section.
- 3 In the **Expression** edit field, type **contactpressure**.
- 4 On the **3D Plot Group** toolbar, click **Plot**.

ID Plot Group 4

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Model Builder** window, under **Results** right-click **ID Plot Group 4** and choose **Rename**.
- 3 Go to the **Rename ID Plot Group** dialog box and type **Displacement** in the **New name** edit field.
- 4 Click **OK**.

Displacement

- 1 On the **ID Plot Group** toolbar, click **Point Graph**.
- 2 Select **Point 70** only.
- 3 In the **Point Graph** settings window, locate the **y-Axis Data** section.
- 4 In the **Expression** edit field, type **w**.
- 5 On the **ID Plot Group** toolbar, click **Plot**.

ID Plot Group 5

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Model Builder** window, under **Results** right-click **ID Plot Group 5** and choose **Rename**.

3 Go to the **Rename ID Plot Group** dialog box and type **Capacitance** in the **New name** edit field.

4 Click **OK**.

Capacitance

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

2 In the **Global** settings window, 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 **ID Plot Group** toolbar, click **Plot**.