

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.

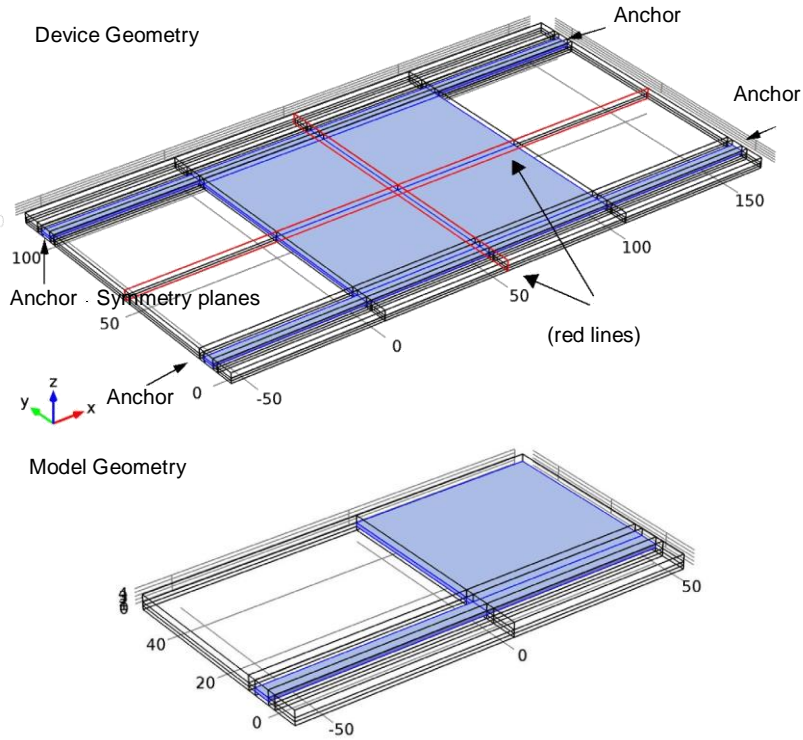


Figure 1: Top: Device geometry showing anchor points and symmetry planes. Bottom: Model geometry. Due to symmetry only one quadrant of the device needs to be modeled.

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

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

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

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.

TODO

1. What are the basic mechanism of the approximate penalty or barrier method.
2. Why was this specific contact force chosen?

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.

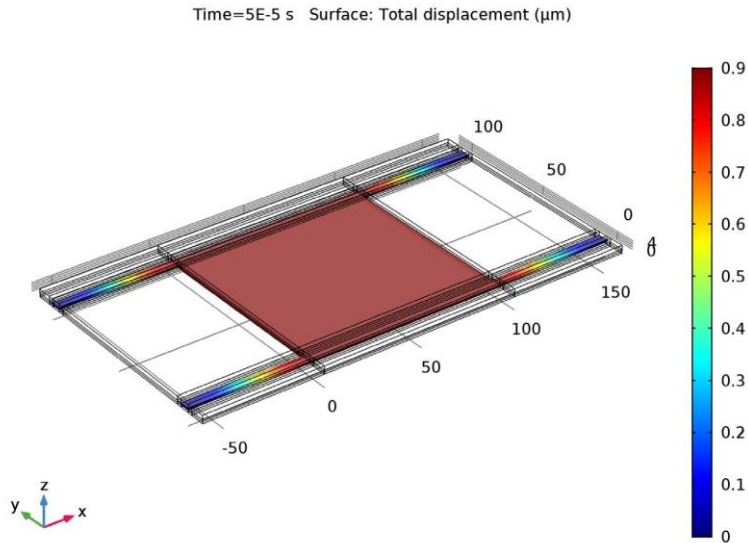


Figure 2: Displacement of the polysilicon when pulled in. Most of the polysilicon structure is in contact with the silicon nitride with a displacement of $0.9 \mu\text{m}$.

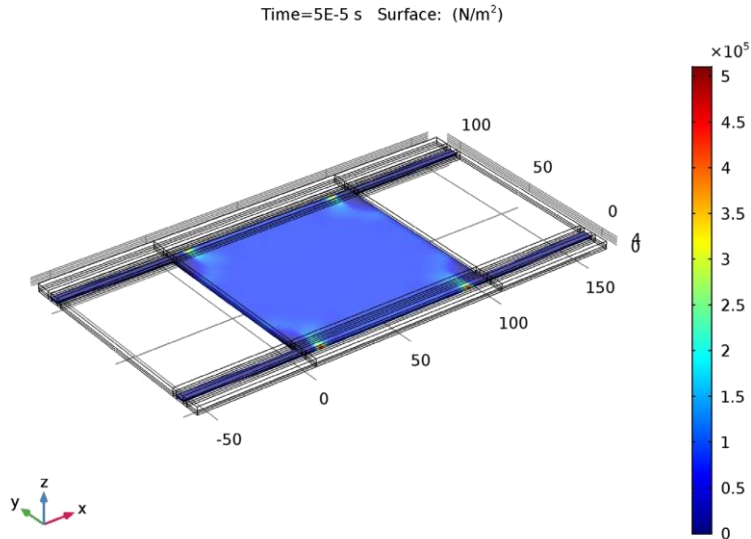


Figure 3: Contact forces acting on the polysilicon when the structure is pulled in.

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.

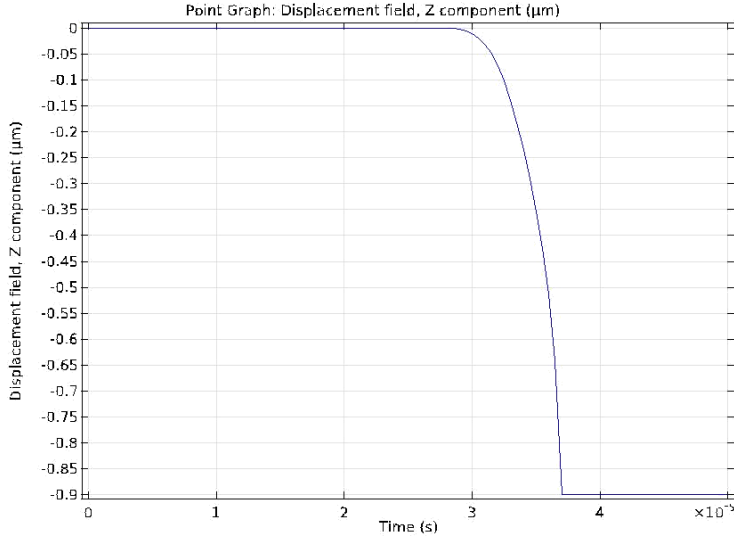


Figure 4: Displacement of the center of the device as a function of time.

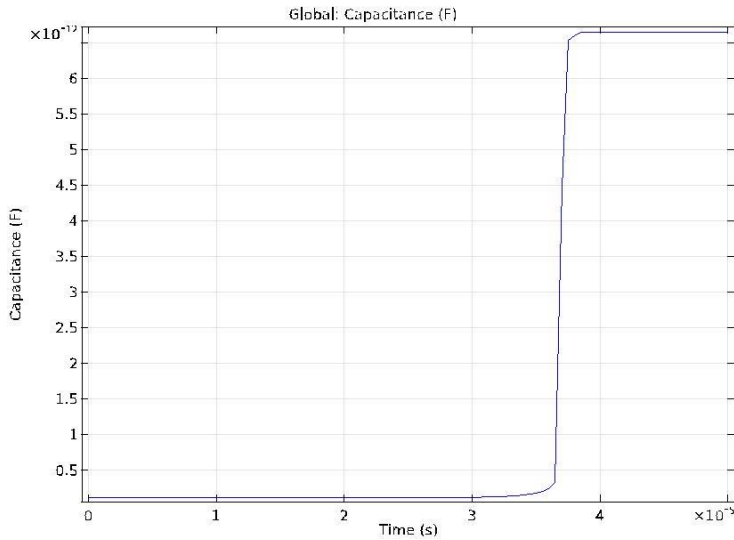


Figure 5: Capacitance of the device as a function of time. The transients in this plot that occur after the point of contact are not physical. The capacitance of the structure changes from 0.1 pF to 1.5 pF as a result of the pull-in.

Reference

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

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/m ³	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/m ²	
Va	V0+Vstep* step2(t/ 1[s])	V	

TODO

1. Gap -> what does the "+w" represent and where does it come from ?
2. Va -> where does the "step2" come from ?
3. Va -> what does the bracket after "step2" mean (t/[s]) ?

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.

4 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 from Plane** section.

3 In the table, enter the following settings:

Distances (μm)
1
2
4

Form Union (fin)

- 1 Right-click **Extrude 1 (ext1)** and choose **Build Selected**.
- 2 On the **Geometry** toolbar, click **Build All**.
- 3 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

DEFINITIONS

TODO:

What are the definitions used for ?

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 $1.05 \cdot \text{insheight}$.
- 4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type $0.05 \cdot \text{insheight}$.

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	epsilon _r	7.5-step1 1 (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 **Settings** window for Linear Elastic Material, 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 **Settings** window for Symmetry, 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 **Settings** window for Symmetry, 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 **Settings** window for Boundary Load, locate the **Force** section.
- 4 Specify the \mathbf{F}_A vector as

0	x
0	y
contactpressure	z

TODO:

1. Why does the z-component of the boundary force has the value “contactpressure”?

Prescribed Mesh Displacement 2

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

Terminal 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Terminal**. Define the voltage applied to the bridge lower side.
- 2 In the **Settings** window for Terminal, 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 text field, type V_a .

Terminal 2

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Terminal**. Define the voltage applied to the base.
- 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 3

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

TODO:

1. **Why is terminal3 needed and what it is used for ?**
- 2 In the **Settings** window for Terminal, 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 text field, type V_0 .

M E S H 1

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

Mapped 1

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

Swept 1

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

Distribution 1

- 1 In the **Settings** window for Distribution, locate the **Domain Selection** section.
- 2 From the **Selection** list, choose **Gap**.
- 3 Locate the **Distribution** section. In the **Number of elements** text field, type 8.
- 4 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

1. Step2: What is the aim of this step?
2. Step2: Why do we have to disable terminal 3
3. Step2: Why do we have to change the default solver?

Time Dependent

On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

Step 2: Time Dependent

- 1 In the **Settings** window for Time Dependent, locate the **Study Settings** section.
- 2 In the **Times** text field, type range(0,5e-7,5e-5).
- 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)>Electromechanics (emi)>Terminal 3**.
- 5 Click **Disable**.

Change the default solver to a **Fully CoupledDirect** solver and use **BDF** time stepping.

TODO: Why do we have to change the default settings to BDF time stepping and Fully Coupled Direct?

Solution 1 (sol1)

TODO: What is time stepping ? and what is used for ?

- 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.

5 Locate the **Time Stepping** section. From the **Method** list, choose **BDF**

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

TODO: What does it mean to set the steps to intermediate ?

7 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** and choose **Fully Coupled**.

Todo: Why do we have to set the time dependent solver 1 to “fully coupled” ? What is it doing?

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

TODO: What is the “method and termination section” used for ?

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

TODO: Why was the “Automatic(Netwon)” chosen ? What is it doing?

10 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Time-Dependent Solver 1** right-click **Direct** and choose **Enable**.

TODO: Why do we have to “enable” the Direct solver ? What is its use ?

11 In the **Settings** window for Direct, locate the **General** section.

12 From the **Solver** list, choose **PARDISO**.

TODO: What is the “PARDISO” and what is it used for ? Why do we need this solver in this specific application ?

13 Click **Compute**.

1