Created in COMSOL Multiphysics 6.2

Pull-In and Pull-Out 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 computes the pull-in and pull-out voltages of the resonator. This is done via a quasistatic analysis of the displacement-voltage trajectory of the full range of motion, from the initial relaxed state all the way to the pulled-in flattened state. The analysis extends the pull-in analysis performed in the accompanying model Pull-In Voltage for 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.

This model computes the pull-in voltage for the resonator by solving an inverse problem. The y-coordinate of a reference point on the resonator is computed using an integration operator (`intop1`). Instead of the midpoint, the reference point is chosen 10 um off-center (Figure 1), so that its y position varies smoothly over the entire range of motion of the cantilever beam. The inverse problem that COMSOL solves computes the DC voltage that must be applied to the beam in order to move the reference point 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, from which the pull-in and pull-out voltages can be read off.

The **Contact** feature is used to simulate the contact between the cantilever beam and the base of the resonator. A small offset is used in the Contact feature to represent a thin oxide layer that prevents the electrode from shorting, and also to prevent mesh collapsing in the air domain in the model.

.

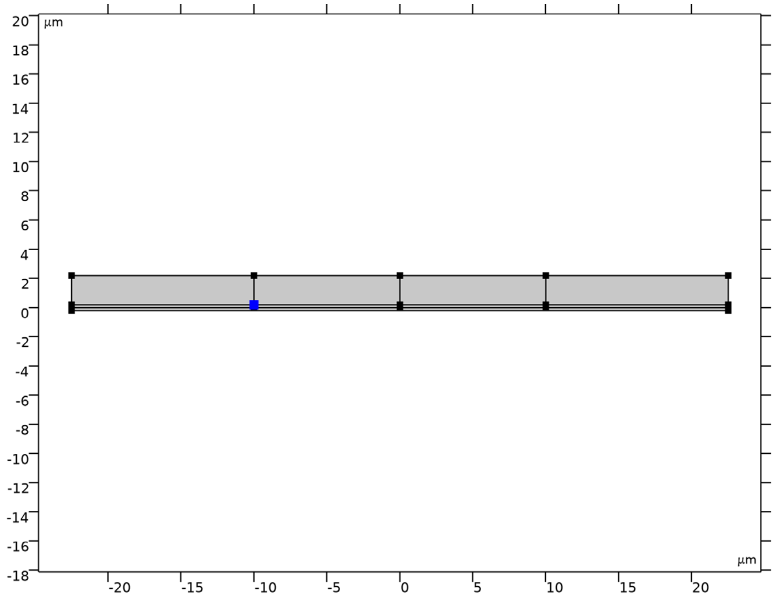


Figure 1: The reference point is chosen 10 um off-center, so that its y position varies smoothly over the entire range of motion of the cantilever beam.

Results and Discussion

Figure 2 shows the deformation of the cantilever beam when it is fully flattened onto the surface of the base of the resonator after pull-in. The aspect ratio of the figure is changed to automatic so that the shape of the flattened beam can be seen easily.

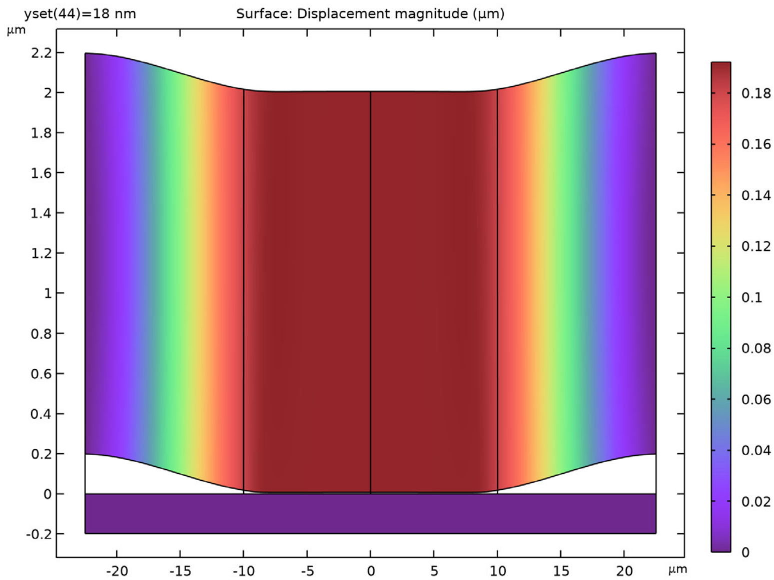


Figure 2: Deformation of the fully flattened beam. The aspect ratio is changed to automatic.

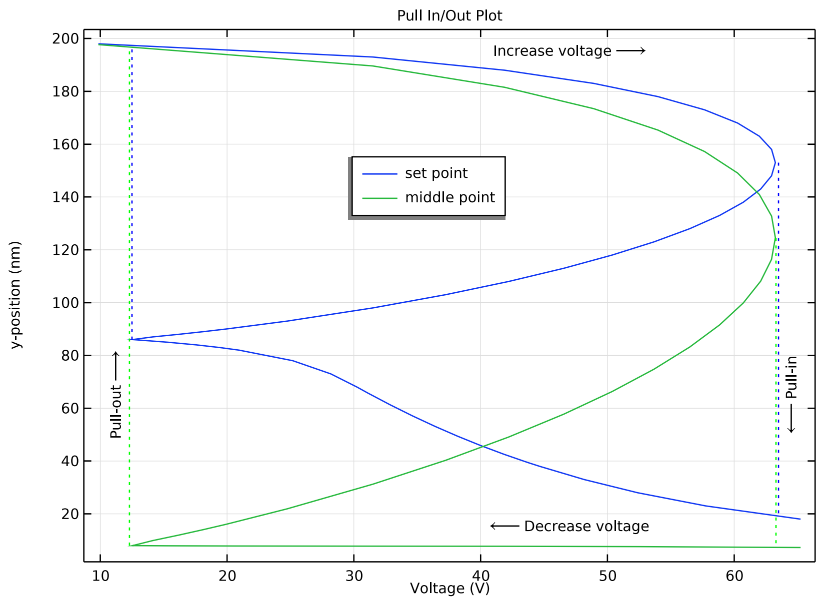


Figure 3: y-coordinates versus voltage over the entire range of motion.

Figure 3 shows the y-coordinates of the reference point (blue) and the center point (green) versus the applied voltage over the entire range of motion of the cantilever beam. The pull-in voltage is about 63 V and the pull-out voltage is about 12.5 V, as can be read off from the figure and indicated by the dotted vertical lines in the figure.

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. Furthermore, the equation is scaled by y_{set} so that it has an overall scale of unity. 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_pull_out`

Modeling Instructions

Start from the existing pull-in model.

APPLICATION LIBRARIES

1. From the **File** menu, choose **Application Libraries**.
2. In the **Application Libraries** window, select **MEMS Module>Actuators>biased_resonator_2d_pull_in** in the tree.
3. Click **Open**.

Create a Union for the Electrostatics domain selection.

GEOMETRY 1

Electrostatics domains

1. In the **Model Builder** window, expand the **Component 1 (comp1)** node.
2. Right-click **Component 1 (comp1)>Geometry 1** and choose **Booleans and Partitions>Union**.
3. In the **Settings** window for **Union**, type `Electrostatics domains` in the **Label** text field.
4. Click in the **Graphics** window and then press Ctrl+A to select all objects.
5. Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Create selections for the electrode boundaries and the destination boundaries of the Contact feature.

Contact destination

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Contact destination` 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 `-11`.

5 In the **x maximum** text field, type `11`.

6 In the **y minimum** text field, type `0.1`.

7 In the **y maximum** text field, type `0.3`.

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

9 Right-click **Contact destination** and choose **Duplicate**.

Electrode

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Contact destination 1 (boxsel2)**.

2 In the **Settings** window for **Box Selection**, type `Electrode` in the **Label** text field.

3 Locate the **Box Limits** section. In the **y minimum** text field, type `-0.1`.

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

We will use the Contact feature to help simulate the configuration of the cantilever beam snapping onto the base of the resonator after pull-in occurs. For the Contact feature to work properly, create a dummy domain for the fixed base of the resonator.

Rectangle 2 (r2)

In the **Model Builder** window, right-click **Rectangle 2 (r2)** and choose **Duplicate**.

Fixed base

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Rectangle 4 (r4)**.

2 In the **Settings** window for **Rectangle**, type `Fixed base` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `22.5*2`.

4 Locate the **Position** section. In the **y** text field, type `-0.1985`.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Create a selection for the source boundaries of the Contact feature.

Electrode (boxsel2)

In the **Model Builder** window, right-click **Electrode (boxsel2)** and choose **Duplicate**.

Contact source

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Electrode 1 (boxsel3)**.

2 In the **Settings** window for **Box Selection**, type `Contact source` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

4 Click **Build Preceding State**.

5 Click **Add**.

6 In the **Add** dialog box, select **Fixed base** in the **Selections** list.

7 Click **OK**.

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

9 In the **x minimum** text field, type `-Inf`.

10 In the **x maximum** text field, type `Inf`.

For the Contact feature to work properly, choose to form an assembly, so that the Electrostatics domains (containing the cantilever beam) and the Fixed base domain remain to be distinct geometric objects.

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**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Clear the **Create pairs** check box.

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

Add a contact pair for the pair of boundaries that will be brought into contact when the cantilever beam is pulled in and flattened onto the surface of the fixed base.

DEFINITIONS

Contact Pair 1 (p1)

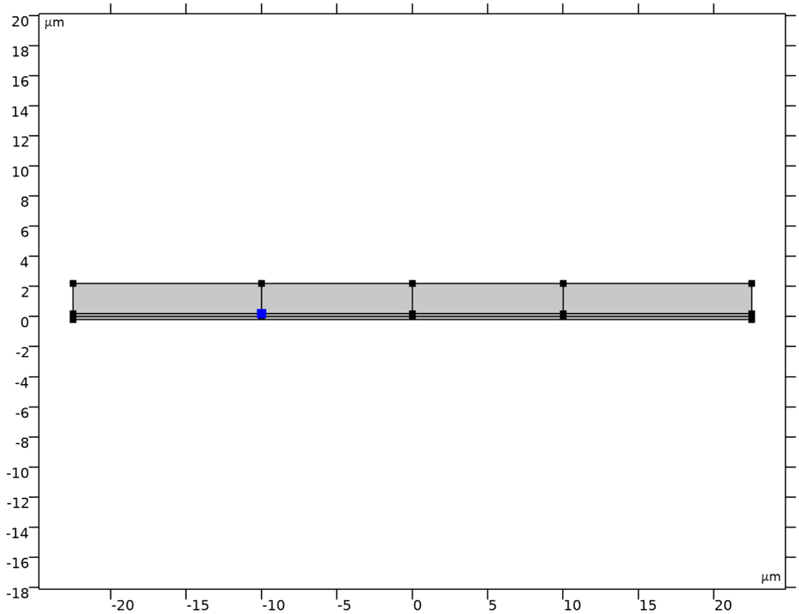
1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.
2 In the **Settings** window for **Pair**, locate the **Source Boundaries** section.
3 From the **Selection** list, choose **Contact source**.
4 Locate the **Destination Boundaries** section. From the **Selection** list, choose **Contact destination**.

For this tutorial, we would like to trace the entire range of motion of the cantilever beam, from the initial relaxed state to the pulled-in fully flattened state. In this case, the middle point of the beam is not a good reference point for the y-position, since it contacts the fixed-base surface first and remains at the same position while the rest of the beam is still going through the pull-in motion. It is better to choose a point off to one side of the beam as the y-position reference, so that its position keeps changing throughout the entire range of motion.

Integration 1 (intop1)

1 In the **Model Builder** window, click **Integration 1 (intop1)**.
2 In the **Settings** window for **Integration**, locate the **Source Selection** section.
3 Click **Clear Selection**.

4 Select Point 9 only.



Add the integration operator for the middle point back, for plotting purposes.

Integration 2 (intop2)

1 In the **Definitions** toolbar, click **Nonlocal 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 12 only.

Add the fixed base domain to the selection of the Solid Mechanics interface. Then make it not moving.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 Select Domains 1, 3, 5, 7, and 9 only.

Fixed Constraint 2

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

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

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

Add a **Contact** feature to the Solid Mechanics interface. Choose the **Penalty** contact method because the default Augmented Lagrangian method is not recommended for fully coupled solvers. Use the **Offset** option to simulate a layer of thin oxide separating the cantilever beam from the fixed-base electrode.

Contact 1a

1 In the **Physics** toolbar, click **Pairs** and choose **Contact**.

2 In the **Settings** window for **Contact**, locate the **Pair Selection** section.

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 1 (p1)** in the **Pairs** list.

5 Click **OK**.

6 In the **Settings** window for **Contact**, click to expand the **Contact Surface Offset and Adjustment** section.

7 In the $d_{\mathrm{offset,d}}$ text field, type `8[nm]`.

Set up Electrostatics feature selections.

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 **Electrostatics domains**.

Electric Potential 1

1 In the **Model Builder** window, expand the **Electrostatics (es)** node, then click **Electric Potential 1**.

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

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

Adjust mesh with a compromise balancing speed, file size, and accuracy. If a very detailed result of the final flattened state is of interest, then a further mesh refinement study is recommended.

MESH 1

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

Distribution 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Mesh 1>Mapped 1** node, then click **Distribution 1**.

2 Select Boundaries 14 and 19 only.

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

4 In the **Number of elements** text field, type `30`.

Distribution 2

1 In the **Model Builder** window, click **Distribution 2**.

2 Select Boundaries 5, 7, 9, and 24 only.

Distribution 3

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

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

3 From the **Selection** list, choose **Contact source**.

4 Locate the **Distribution** section. In the **Number of elements** text field, type `50`.

5 Click **Build All**.

Extend the range of Auxiliary Sweep to cover the entire range of motion of the beam.

PULL IN - FULL RANGE

1 In the **Model Builder** window, click **Pull In**.

2 In the **Settings** window for **Study**, type `Pull In - Full Range` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, expand the **Pull In - Full Range** node, then click **Step 1: Stationary**.

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

3 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit

yset (Set point y-coordinate)	`range(198, -5, 93)`
	`range(90, -1, 82)`
	`range(78, -5, 18)`
		nm

Compute.

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

RESULTS

Pull In Displacement

Change the aspect ratio of the displacement plot to show in detail the fully pulled-in flattened state.

1 In the **Settings** window for **2D Plot Group**, locate the **Plot Settings** section.
2 From the **View** list, choose **New view**.
3 In the **Pull In Displacement** toolbar, click **Plot**.
4 Click **Go to Source**.

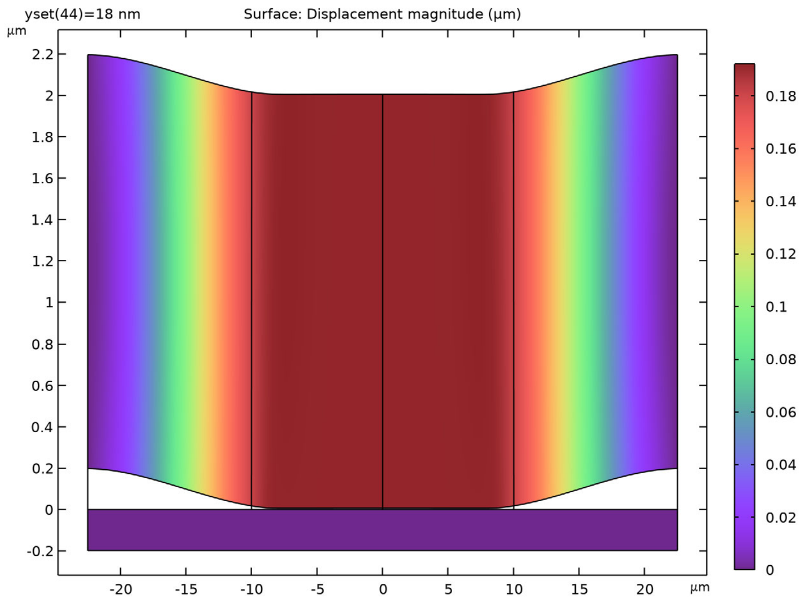
Axis

1 In the **Model Builder** window, expand the **View 2D 2** 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 in the **Graphics** toolbar.

Pull In Displacement

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

2 In the **Settings** window for **2D Plot Group**, click **Plot Last**.



Flip the axes of the Pull In Plot so that the vertical position (y) is plotted on the vertical axis, and the voltage is plotted on the horizontal axis.

Pull In/Out Plot

1 In the **Model Builder** window, under **Results** click **Pull In Plot**.

2 In the **Settings** window for **1D Plot Group**, type `Pull In/Out Plot` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Pull In/Out Plot`.

5 Locate the **Plot Settings** section.

6 Select the **y-axis label** check box. In the associated text field, type `y-position (nm)`.

Global 1

1 In the **Model Builder** window, expand the **Pull In/Out Plot** node, then click **Global 1**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`yset`	`nm`	`set point`
`intop2(y)`	`nm`	`middle point`

4 Locate the **x-Axis Data** section. In the **Expression** text field, type `abs(VdcSP)[V]`.

5 Select the **Description** check box. In the associated text field, type `Voltage`.

Add annotations to indicate the pull-in and pull-out events. Enter the coordinates of the pull-in and pull-out transitions into a text file. Then import the file into a table to plot the transitions using Table Graphs.

Table 1

1 In the **Results** toolbar, click **Table**.

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

3 Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `biased_resonator_2d_pull_in_pull_out_table.txt`.

Pull In/Out Plot

1 In the **Model Builder** window, under **Results** click **Pull In/Out Plot**.

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

3 Select the **x-axis label** check box.

4 Locate the **Legend** section. From the **Position** list, choose **Manual**.

5 In the **x-position** text field, type `30/65`.

6 In the **y-position** text field, type `150/200`.

Table Graph 1

1 Right-click **Pull In/Out Plot** and choose **Table Graph**.

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

3 From the **x-axis data** list, choose **Column 1**.

4 From the **Plot columns** list, choose **Manual**.

5 In the **Columns** list, select **Column 2**.

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

7 From the **Color** list, choose **Blue**.

8 Right-click **Table Graph 1** and choose **Duplicate**.

Table Graph 2

1 In the **Model Builder** window, click **Table Graph 2**.

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

3 From the **x-axis data** list, choose **Column 3**.

4 In the **Columns** list, select **Column 4**.

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

6 Right-click **Table Graph 2** and choose **Duplicate**.

Table Graph 3

1 In the **Model Builder** window, click **Table Graph 3**.

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

3 From the **x-axis data** list, choose **Column 5**.

4 In the **Columns** list, select **Column 6**.

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

6 Right-click **Table Graph 3** and choose **Duplicate**.

Table Graph 4

1 In the **Model Builder** window, click **Table Graph 4**.

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

3 From the **x-axis data** list, choose **Column 7**.

4 In the **Columns** list, select **Column 8**.

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

Annotation 1

1 In the **Model Builder** window, right-click **Pull In/Out Plot** and choose **Annotation**.

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

3 In the **Text** text field, type `$\longleftarrow$  Pull-in`.

4 Locate the **Position** section. In the **x** text field, type `64.5`.

5 In the **y** text field, type `65`.

6 Locate the **Annotation** section. Select the **LaTeX markup** check box.

7 Locate the **Coloring and Style** section. Clear the **Show point** check box.

8 From the **Anchor point** list, choose **Center**.

9 From the **Orientation** list, choose **Vertical**.

10 Right-click **Annotation 1** and choose **Duplicate**.

Annotation 2

1. In the **Model Builder** window, click **Annotation 2**.
2. In the **Settings** window for **Annotation**, locate the **Annotation** section.
3. In the **Text** text field, type `Pull-out  $\longrightarrow$`.
4. Locate the **Position** section. In the **x** text field, type `11.2`.
5. Right-click **Annotation 2** and choose **Duplicate**.

Annotation 3

1. In the **Model Builder** window, click **Annotation 3**.
2. In the **Settings** window for **Annotation**, locate the **Annotation** section.
3. In the **Text** text field, type `Increase voltage  $\longrightarrow$`.
4. Locate the **Position** section. In the **x** text field, type `47`.
5. In the **y** text field, type `195`.
6. Locate the **Coloring and Style** section. From the **Orientation** list, choose **Horizontal**.
7. Right-click **Annotation 3** and choose **Duplicate**.

Annotation 4

1. In the **Model Builder** window, click **Annotation 4**.
2. In the **Settings** window for **Annotation**, locate the **Annotation** section.
3. In the **Text** text field, type `$\longleftarrow$  Decrease voltage`.
4. Locate the **Position** section. In the **y** text field, type `15`.

5 In the **Pull In/Out Plot** toolbar, click **Plot**.