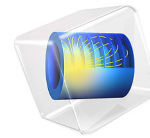
Created in COMSOL Multiphysics 5.2a



Piezoceramic Tube

Introduction

This example involves a static 2D axisymmetric analysis of a piezoelectric actuator using the Piezoelectric Devices multiphysics interface. It models a radially polarized piezoelectric tube, as described by S. Peelamedu and co-authors (Ref. 1). An application area where radially polarized tubes are employed is in nozzles for fluid control in inkjet printers.

Model Definition

GEOMETRY

The tube has a height of 0.62 mm and an inner and outer radius of 0.38 mm and 0.62 mm, respectively. It is represented in an axisymmetric geometry by a single off-axis rectangle, as shown in Figure 1.

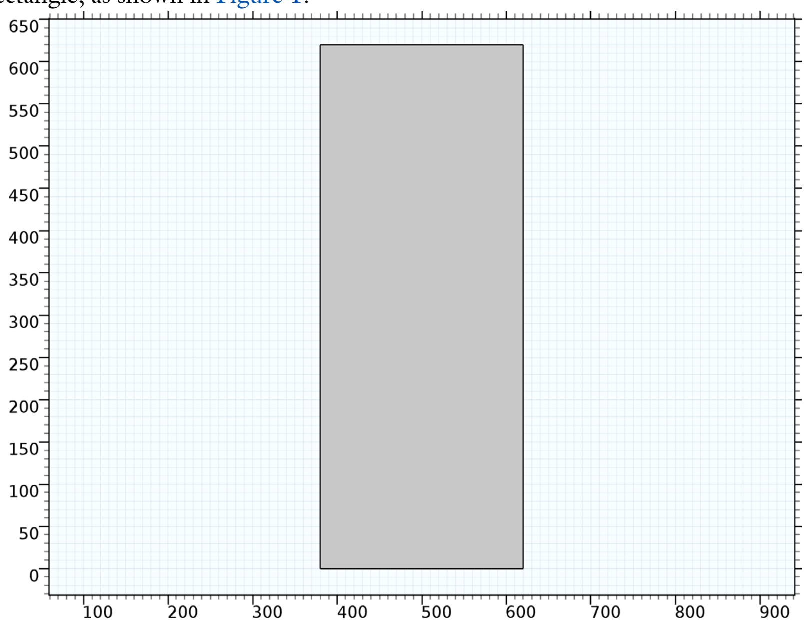


Figure 1: The axisymmetric geometry. Length units on x- and y-axes are shown in µm.

BOUNDARY CONDITIONS

The model studies two cases distinguished by different boundary conditions. Case 1 represents the direct piezoelectric effect, and Case 2 represents the inverse piezoelectric effect.

Case 1—Direct Piezoelectric Effect:

- Structural mechanics boundary condition—constrain the bottom surface from moving axially (in the z direction), but also add an internal fluid pressure of 0.1 MPa.
- Electrostatics boundary condition—ground the inner and outer surfaces.

Case 2—Inverse Piezoelectric Effect:

- Structural mechanics boundary condition—constrain the bottom surface from moving axially (in the z-direction).
- Electrostatics boundary condition—apply a 1 V potential difference between the tube's inner and outer surfaces.

MATERIAL ORIENTATION

COMSOL's material library data is entered in a form which assumes that the crystal polarization is aligned with the global co-ordinate z axis. For the radially polarized case treated in this model, the orientation must be rotated so that the material polarization direction is aligned with the r direction (radially polarized). To do so, specify the co-ordinate system in the Piezoelectric Material feature. By selecting the co-ordinate system as the predefined zx-plane system, you rotate the material so that its z direction is aligned with the r direction of the model, and the material's x direction is aligned with the model's z direction.

The piezoceramic material in this example (PZT-5H) is a transversely isotropic material, which is a special class of orthotropic materials. Such a material has the same properties in one plane (isotropic behavior) and different properties in the direction normal to this plane. Thus you can use either the zx-plane material orientation or the zy-plane material orientation; both give the same solution.

Results and Discussion

Figure 2 shows the radial displacement due to the applied pressure in Case 1, and Figure 3 shows the corresponding induced electric potential. Both the radial displacement and potential are shown along a cut line 300 μm above the base of the tube in Figure 4 and Figure 5, receptively.

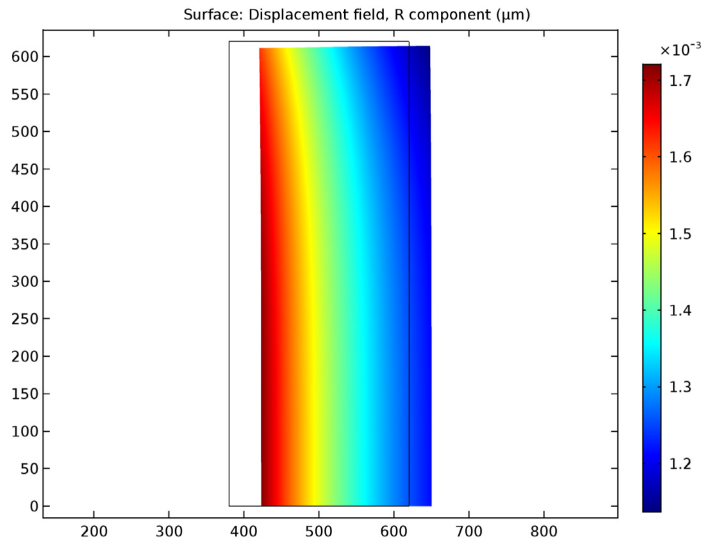


Figure 2: Deformed shape and radial displacement due to an internal pressure of 0.1 MPa (case 1 —the direct piezoelectric effect).

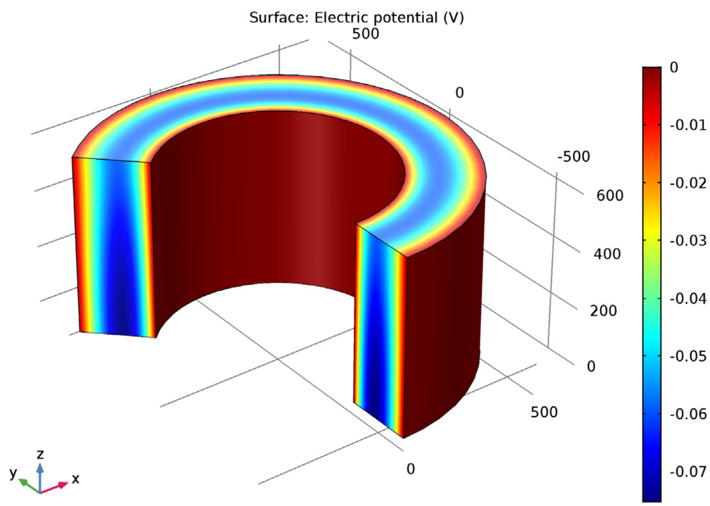


Figure 3: Induced electric potential within the deformed tube due to an internal pressure of 0.1 MPa (case 1 —the direct piezoelectric effect).

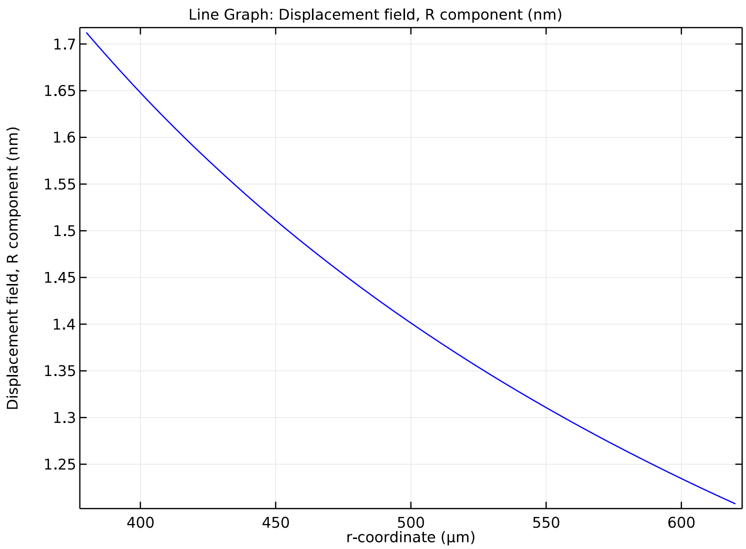


Figure 4: Radial displacement as a function of r-coordinate at a height of 300 µm above the base of the tube. The results are for Case 1—the direct piezoelectric effect.

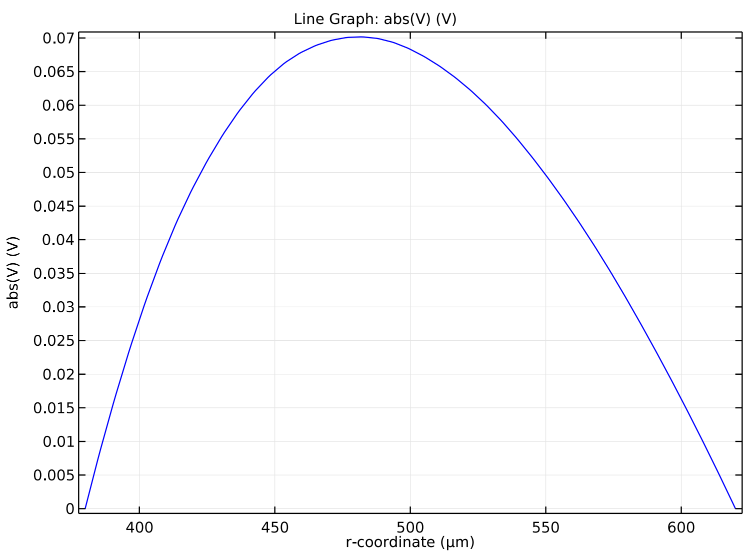


Figure 5: Electric potential as a function of r-coordinate at a height of 300 µm above the base of the tube. The results are for Case 1—the direct piezoelectric effect.

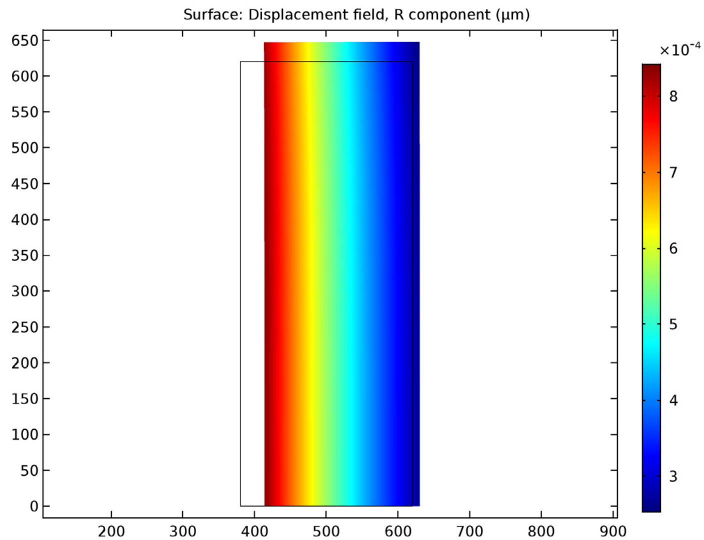


Figure 6: Deformed shape and radial displacement of the piezoceramic-tube actuator due to the radial electric field (Case 2—Inverse Piezoelectric Effect).

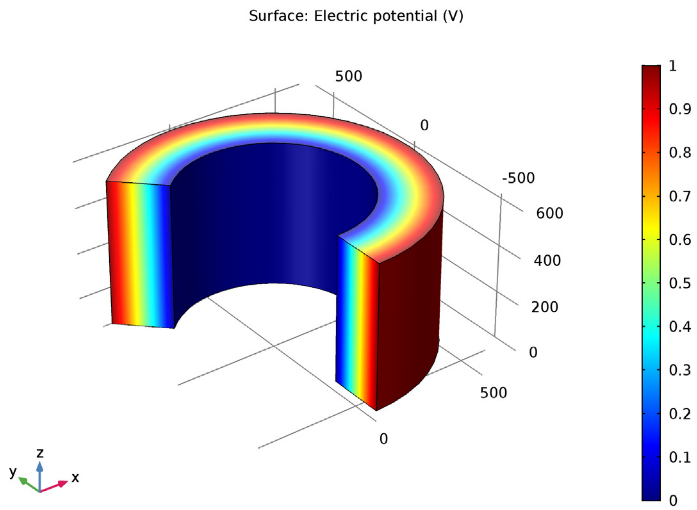


Figure 7: Electric potential applied to the tube to induce the displacements shown in Figure 6 *(Case 2—Inverse Piezoelectric Effect).*

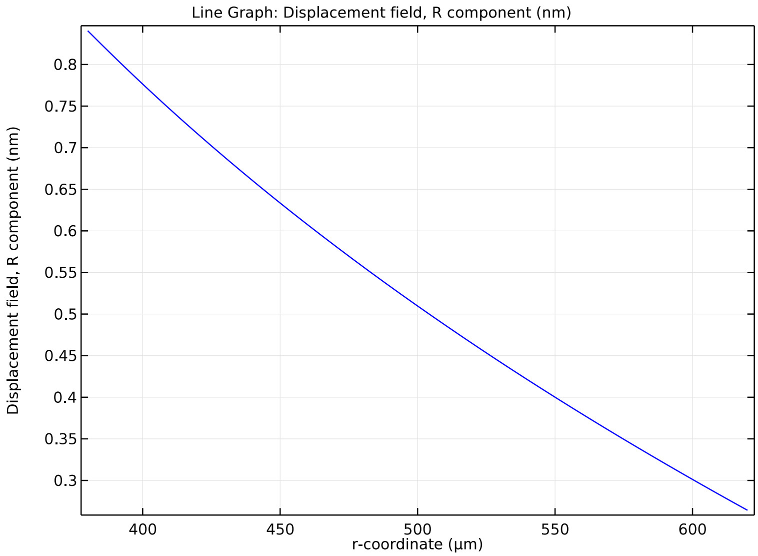


Figure 8: Radial displacement as a function of r-coordinate at a height of 300 µm above the base of the tube. The results are for Case 2—the inverse piezoelectric effect.

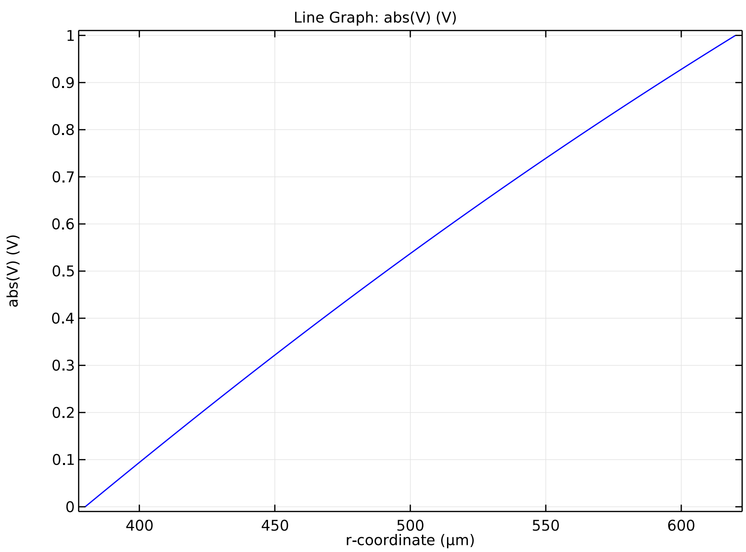


Figure 9: Electric potential as a function of r-coordinate at a height of 300 µm above the base of the tube. The results are for Case 2—the inverse piezoelectric effect.

Figure 6 shows the radial displacement resulting from the applied potential shown in Figure 7. The radial displacement and potential are shown along a cut line 300 µm above the base of the tube in Figure 8 and Figure 9, respectively.

These results show good agreement with those from S. Peelamedu (Ref. 1).

Reference

1. S.M. Peelamedu, C.B. Kosaraju, R.V. Dukkipati and N.G. Naganathan, “Numerical Approach for Axisymmetric Piezoceramic Geometries towards Fluid Control Applications, ” *Proceedings of the Institution of Mechanical Engineers, Part I: J. Systems and Control Engineering*, vol. 214, no. 2, pp. 87–97, 2000.

Application Library path: `MEMS_Module/Piezoelectric_Devices/piezoceramic_tube`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1. In the **Model Wizard** window, click **2D Axisymmetric**.
2. In the **Select Physics** tree, select **Structural Mechanics>Piezoelectric Devices**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
6. Click **Done**.

GEOMETRY 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2. In the **Settings** window for Geometry, locate the **Units** section.

3 From the **Length unit** list, choose **µm**.

Create the tube by adding an off-axis rectangle in the axisymmetric geometry.

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type 240.

4 In the **Height** text field, type 620.

5 Locate the **Position** section. In the **r** text field, type 380.

6 Click **Build All Objects**.

Add a PZT 5H to the model.

ADD MATERIAL

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

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

3 In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-5H)**.

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

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

SOLID MECHANICS (SOLID)

Piezoelectric Material 1

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

2 In the **Settings** window for Piezoelectric Material, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Material ZX-Plane System (comp1_zx_sys)**.

By selecting the material orientation as the zx-plane, you rotate the material so that its z direction is aligned with the r direction of the model, and the material's x direction is aligned with the model's z direction.

This example comprises two studies: the direct effect and inverse effect. All loadings for both studies are defined together and then a selection of relevant features will be done in the study settings.

Add a pressure follower load to the inner surface of the cylinder.

Boundary Load 1

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

2 Select Boundary 1 only.

3 In the **Settings** window for Boundary Load, locate the **Force** section.

4 From the **Load type** list, choose **Pressure**.

5 In the p text field, type `0.1[MPa]`.

Constrain the lower surface of the tube with a roller boundary condition.

Roller 1

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

2 Select Boundary 2 only.

ELECTROSTATICS (ES)

Ground both the inner and outer surfaces of the cylinder.

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

Ground 1

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

2 Select Boundaries 1 and 4 only.

Add an electric potential feature on the outer boundary. This will over-ride the existing **Ground** feature.

Electric Potential 1

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

2 Select Boundary 4 only.

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

4 In the V_0 text field, type `1`.

MESH 1

Create a mapped mesh.

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

Mapped 1

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

STUDY 1

The first study simulates the direct effect. All mechanical loads are kept and the electric potential feature is disabled in solver settings. It is automatically replaced by the ground feature that was previously overridden.

Step 1: Stationary

1. In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2. In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.
3. Select the **Modify physics tree and variables for study step** check box.
4. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Electrostatics (es)>Electric Potential 1**.
5. Click **Disable**.
6. On the **Home** toolbar, click **Compute**.

RESULTS

Stress (solid)

The default plot groups show stress in the tube and the induced electric potential. Adapt these for comparison with Ref. 1. First replace stress plot by radial displacement.

1. In the **Model Builder** window, under **Results** click **Stress (solid)**.
2. In the **Settings** window for 2D Plot Group, type `Radial Displacement (Direct Effect)` in the **Label** text field.

Surface 1

1. In the **Model Builder** window, expand the **Results>Radial Displacement (Direct Effect)** node, then click **Surface 1**.
2. In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Displacement>Displacement field (Material)>u - Displacement field, R component**.
3. On the **Radial Displacement (Direct Effect)** toolbar, click **Plot**.

Stress, 3D (solid)

1. In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.
2. In the **Settings** window for 3D Plot Group, type `Stress, 3D (Direct Effect)` in the **Label** text field.

Electric Potential (es)

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

2 In the **Settings** window for 2D Plot Group, type `Electric Potential (Direct Effect)` in the **Label** text field.

Electric Potential, Revolved Geometry (es)

Change the data set of the potential plot in order to see a 3D cut view of the potential.

1 In the **Model Builder** window, under **Results** click **Electric Potential, Revolved Geometry (es)**.

2 In the **Settings** window for 3D Plot Group, type `Electric Potential, 3D (Direct Effect)` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Revolution 2D 1**.

4 On the **Electric Potential, 3D (Direct Effect)** toolbar, click **Plot**.

Data Sets

Create a cross section through the geometry to use for line plots of the electric potential and displacement.

Cut Line 2D 1

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for Cut Line 2D, locate the **Line Data** section.

3 In row **Point 1**, set **r** to `380` and **z** to `300`.

4 In row **Point 2**, set **r** to `620` and **z** to `300`.

Visualize the cross section line.

5 Click **Plot**.

Add line plots of the radial displacement and the potential along the cross section.

1D Plot Group 5

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Radial Displacement, cut (Direct Effect)` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 1**.

Line Graph 1

1 On the **Radial Displacement, cut (Direct Effect)** toolbar, click **Line Graph**.

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

3 In the **Settings** window for Line Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Displacement>Displacement field (Material)>u - Displacement field, R component**.

4 Locate the **y-Axis Data** section. From the **Unit** list, choose **nm**.

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

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

7 On the **Radial Displacement, cut (Direct Effect)** toolbar, click **Plot**.

Radial Displacement, cut (Direct Effect)

In the **Model Builder** window, under **Results** right-click **Radial Displacement, cut (Direct Effect)** and choose **Duplicate**.

Radial Displacement, cut (Direct Effect) 1

In the **Settings** window for 1D Plot Group, type `Voltage, Cut (Direct Effect)` in the **Label** text field.

Line Graph 1

1 In the **Model Builder** window, expand the **Results>Voltage, Cut (Direct Effect)** node, then click **Line Graph 1**.

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

3 In the **Expression** text field, type `abs(V)`.

4 On the **Voltage, Cut (Direct Effect)** toolbar, click **Plot**.

Finally add a new study to compute the results for the inverse effect.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

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

STUDY 2

The second study simulates the inverse effect. All electrical loads are kept and the pressure load feature is disabled in solver settings. It is automatically replaced by the **Free** boundary feature that was previously overridden.

Step 1: Stationary

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

2 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Boundary Load 1**.
5 Click **Disable**.
6 On the **Home** toolbar, click **Compute**.

RESULTS

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
2 In the **Settings** window for 2D Plot Group, type `Radial Displacement (Inverse Effect)` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Radial Displacement (Inverse Effect)** node, then click **Surface 1**.
2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics> Displacement>Displacement field (Material)>u - Displacement field, R component**.
3 On the **Radial Displacement (Inverse Effect)** toolbar, click **Plot**.

Stress, 3D (solid)

1 In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.
2 In the **Settings** window for 3D Plot Group, type `Stress, 3D (Inverse Effect)` in the **Label** text field.

Electric Potential (es)

1 In the **Model Builder** window, under **Results** click **Electric Potential (es)**.
2 In the **Settings** window for 2D Plot Group, type `Electric Potential (Inverse Effect)` in the **Label** text field.

Electric Potential, Revolved Geometry (es)

1 In the **Model Builder** window, under **Results** click **Electric Potential, Revolved Geometry (es)**.
2 In the **Settings** window for 3D Plot Group, locate the **Data** section.
3 From the **Data set** list, choose **Revolution 2D 3**.
4 In the **Label** text field, type `Electric Potential, 3D (Inverse Effect)`.
5 On the **Electric Potential, 3D (Inverse Effect)** toolbar, click **Plot**.

Create a second **Cut Line 2D** for the new solution.

Cut Line 2D 1

In the **Model Builder** window, under **Results>Data Sets** right-click **Cut Line 2D 1** and choose **Duplicate**.

Cut Line 2D 2

1 In the **Settings** window for Cut Line 2D, locate the **Data** section.

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

Radial Displacement, cut (Direct Effect)

In the **Model Builder** window, under **Results** right-click **Radial Displacement, cut (Direct Effect)** and choose **Duplicate**.

Radial Displacement, cut (Direct Effect) 1

1 In the **Settings** window for 1D Plot Group, type `Radial Displacement, cut (Inverse Effect)` in the **Label** text field.

2 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 2**.

3 On the **Radial Displacement, cut (Inverse Effect)** toolbar, click **Plot**.

Voltage, Cut (Direct Effect)

In the **Model Builder** window, under **Results** right-click **Voltage, Cut (Direct Effect)** and choose **Duplicate**.

Voltage, Cut (Direct Effect) 1

1 In the **Settings** window for 1D Plot Group, type `Voltage, Cut (Inverse Effect)` in the **Label** text field.

2 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 2**.

3 On the **Voltage, Cut (Inverse Effect)** toolbar, click **Plot**.