Created in COMSOL Multiphysics 6.0

# Microresistor Beam

## Introduction

This example illustrates the ability to couple thermal, electrical, and structural analysis in one model. This particular application moves a beam by passing a current through it; the current generates heat, and the temperature increase leads to displacement through thermal expansion. The model estimates how much current and increase in temperature are necessary to displace the beam.

Although the model involves a rather simple 3D geometry and straightforward physics, it provides a good example of multiphysics modeling.

## Model Definition

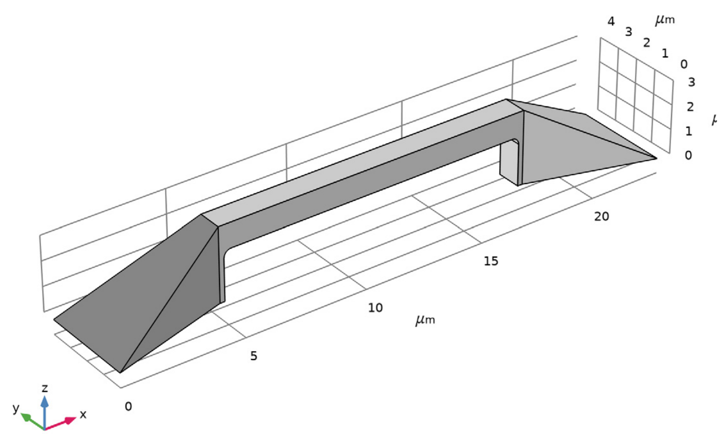



*Figure 1: Microbeam geometry.*

A copper microbeam has a length of 13 µm with a height and width of 1 µm. Feet at both ends bond it rigidly to a substrate. An electric potential of 0.2 V applied between the feet induces an electric current. Due to the material's resistivity, the current heats up the structure. Because the beam operates in the open, the generated heat dissipates into the air. The thermally induced stress loads the material and deforms the beam.

As a first approximation, you can assume that the electrical conductivity is constant. However, a conductor's resistivity increases with temperature. In the case of copper, the relationship between resistivity and temperature is approximately linear over a wide range of temperatures:

$$\rho = \rho_0(1 + \alpha(T - T_0)) \quad (1)$$

$\alpha$ is the temperature coefficient. You obtain the conductor's temperature dependency from the relationship that defines electric resistivity; conductivity is simply its reciprocal ($\sigma = 1/\rho$).

For the heat transfer equations, set the base boundaries facing the substrate to a constant temperature of 323 K. You model the convective air cooling in other boundaries using a heat flux boundary condition with a heat transfer coefficient, $h$, of 5 W/(m$^2$·K) and an external temperature, $T_{inf}$, of 298 K. Standard constraints handle the bases' rigid connection to the substrate.

## *Results and Discussion*

Figure 2 shows the temperature field on the microbeam surface when solving the model using a temperature-dependent resistivity as in Equation 1. Based on the color scale, the maximum temperature is about 710 K.

Figure 3 shows the microbeam's deformation. The displacement for the temperature-dependent case is 48 nm compared to the maximum displacement for constant electrical conductivity, which is 88 nm (the plot scales the deformation by a factor of around 20).

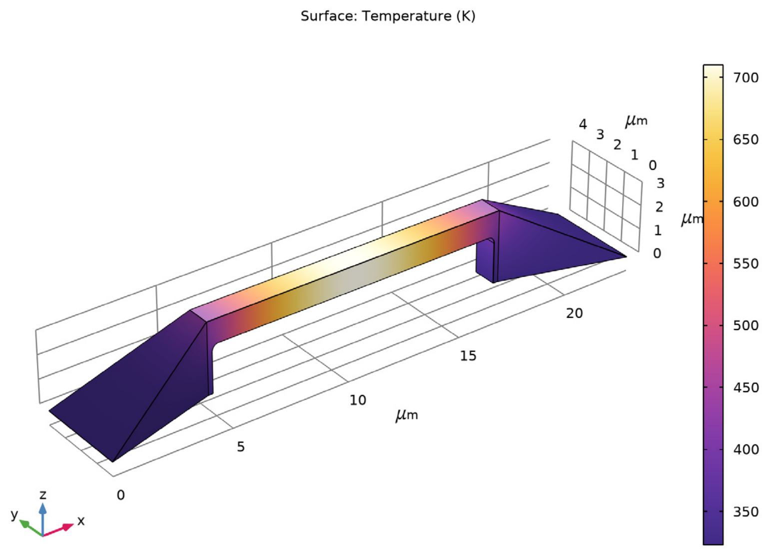



*Figure 2: Surface temperature with temperature-dependent electrical conductivity.*

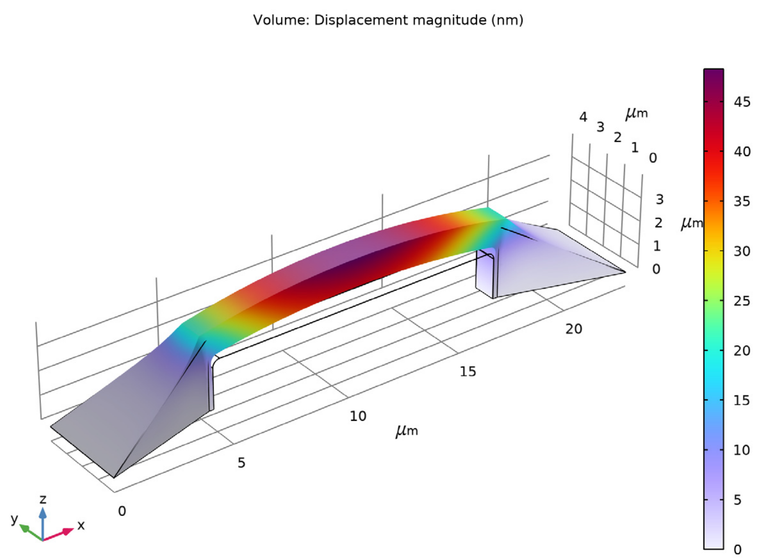



*Figure 3: Microbeam deformation with temperature-dependent electrical conductivity.*

## *Notes About the COMSOL Implementation*

In this example you create the 3D geometry by starting with two 2D work planes. The first one views the geometry from above, and the second does so from the side. You create cross sections on the work planes, which you then extrude into 3D. As the final step you create the resistor beam geometry as the intersection of the extruded objects. You can also skip the step-by-step instructions for the geometry creation and import the ready-made geometry directly from the Application Libraries.

By using the *Joule Heating and Thermal Expansion* predefined multiphysics interface you automatically add the equations for three physics including the necessary multiphysics couplings. In this case the physics equations describe the current and heat conduction and structural mechanics problems. The interface also provides suitable defaults for the solver.

---

**Application Library path:** `MEMS_Module/Actuators/microresistor_beam`

---

## *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>Thermal-Structure Interaction>Joule Heating and Thermal Expansion**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

### 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 | 0.2[V] | 0.2 V | Applied voltage |
| T0 | 323[K] | 323 K | Heat sink temperature |
| Text | 298[K] | 298 K | External temperature |
| k | 5[W/(m^2*K)] | 5 W/(m²·K) | Heat transfer coefficient |

## 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**.

### *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)>Polygon 1 (pol1)*

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

**2** In the **Settings** window for **Polygon**, locate the **Coordinates** section.

**3** From the **Data source** list, choose **Vectors**.

**4** In the **xw** text field, type `0 5 5 18 18 23 23 23 23 18 18 5 5 0 0 0`.

**5** In the **yw** text field, type `0 1.5 1.5 1.5 1.5 0 0 4 4 2.5 2.5 2.5 2.5 4 4 0`.

### *Extrude 1 (ext1)*

**1** In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Extrude**.

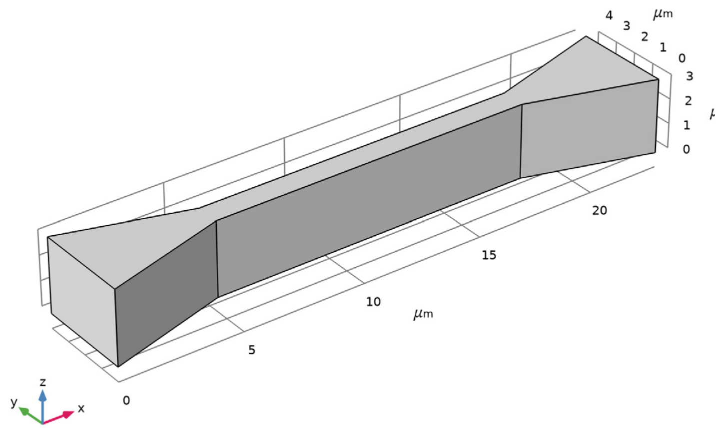
**2** In the **Settings** window for **Extrude**, locate the **Distances** section.

**3** In the table, enter the following settings:

| Distances (µm) |
|---|
| 3 |

**4** Click **Build All Objects**.

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




*Work Plane 2 (wp2)*

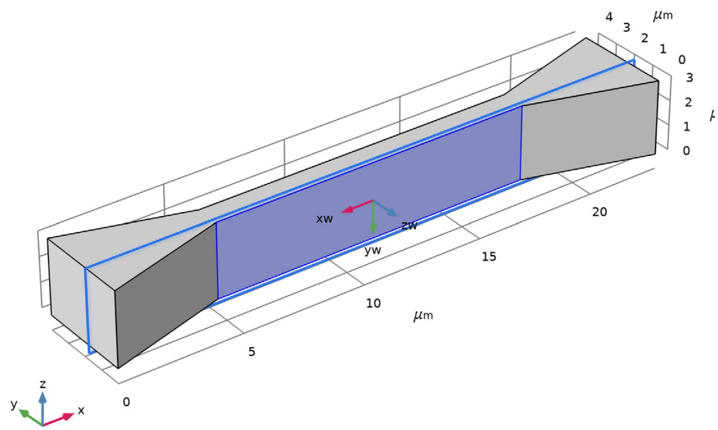
**1** In the **Geometry** toolbar, click **Work Plane**.

**2** In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

**3** From the **Plane type** list, choose **Face parallel**.

**4** On the object **ext1**, select Boundary 6 only.

It might be easier to select the correct boundary by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)




**5** In the **Offset in normal direction** text field, type `-1.5`.

**6** Select the **Reverse normal direction** check box.

**7** Click **Show Work Plane**.

*Work Plane 2 (wp2)>Plane Geometry*

**1** In the **Settings** window for **Plane Geometry**, locate the **Visualization** section.

**2** Find the **In-plane visualization of 3D geometry** subsection. Clear the **Intersection (green)** check box.

**3** Clear the **Coincident entities (blue)** check box.

*Work Plane 2 (wp2)>Polygon 1 (pol1)*

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

**2** In the **Settings** window for **Polygon**, locate the **Coordinates** section.

**3** From the **Data source** list, choose **Vectors**.

**4** In the **xw** text field, type `-11.5 -6.3 -6.3 -6.3 -6.3 6.3 6.3 6.3 6.3 11.5 11.5 6.5 6.5 -6.5 -6.5 -11.5`.

5 In the **yw** text field, type `-1.5 -1.5 -1.5 0.5 0.5 0.5 0.5 -1.5 -1.5 -1.5 -1.5 1.5 1.5 1.5 1.5 -1.5`.

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

*Work Plane 2 (wp2)>Fillet 1 (fil1)*

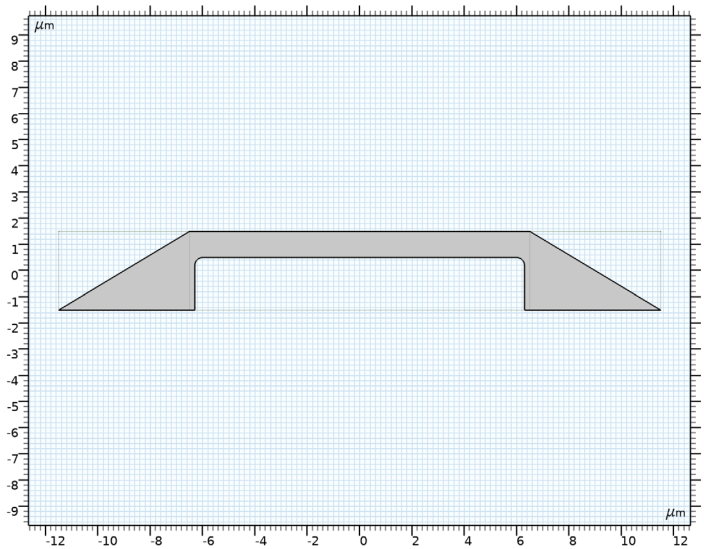
1 In the **Work Plane** toolbar, click **Fillet**.

2 On the object **pol1**, select Points 4 and 6 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `0.3`.

5 In the **Work Plane** toolbar, click **Build All**.




*Extrude 2 (ext2)*

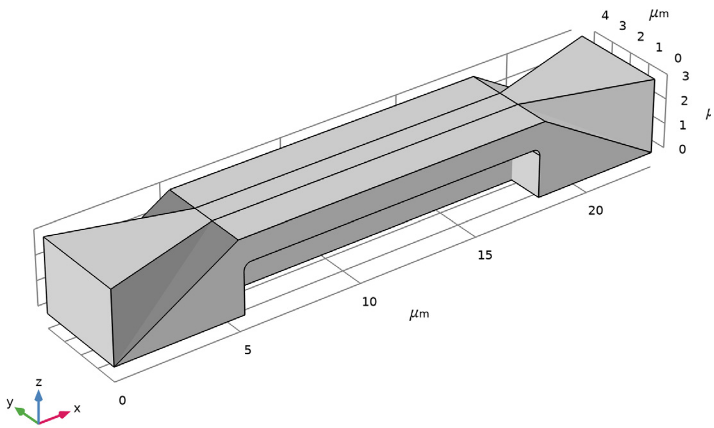
1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 2 (wp2)** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

| Distances (μm) |
| --- |
| 4 |

4 Click **Build All Objects**.




### *Intersection 1 (int1)*

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Intersection**, click **Build All Objects**.

### *Form Union (fin)*

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

The model geometry is now complete.

## DEFINITIONS

Add a set of selections that you can use later when applying boundary conditions.

### *connector1*

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

2 Right-click **Explicit 1** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `connector1` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

6 From the **Geometric entity level** list, choose **Boundary**.

7 Select Boundary 1 only.

### connector2

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

2 In the **Model Builder** window, right-click **Explicit 2** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `connector2` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

6 From the **Geometric entity level** list, choose **Boundary**.

7 Select Boundary 13 only.

### connectors

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

2 Right-click **Explicit 3** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `connectors` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

6 From the **Geometric entity level** list, choose **Boundary**.

7 Select Boundaries 1 and 13 only.

## 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 **MEMS>Metals>Cu - Copper**.

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

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

## MATERIALS

### *Cu - Copper (mat1)*

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

**2** In the table, enter the following settings:

| **Property** | **Variable** | **Value** | **Unit** | **Property group** |
|---|---|---|---|---|
| Relative permittivity | epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0 | 1 | 1 | Basic |

**3** Click to expand the **Material Properties** section. In the **Material properties** tree, select **Electromagnetic Models>Linearized Resistivity>Reference resistivity (rho0)**.

**4** Click **Add to Material**.

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

| **Property** | **Variable** | **Value** | **Unit** | **Property group** |
|---|---|---|---|---|
| Reference resistivity | rho0 | `1.72e-8[ohm*m]` | Ω·m | Linearized resistivity |
| Resistivity temperature coefficient | alpha | `0.0039[1/K]` | 1/K | Linearized resistivity |
| Reference temperature | Tref | `293[K]` | K | Linearized resistivity |

## ELECTRIC CURRENTS (EC)

### *Current Conservation 1*

**1** In the **Model Builder** window, under **Component 1 (comp1)>Electric Currents (ec)** click **Current Conservation 1**.

**2** In the **Settings** window for **Current Conservation**, locate the **Constitutive Relation Jc-E** section.

**3** From the **Conduction model** list, choose **Linearized resistivity**.

Before solving the bidirectionally coupled model with a temperature-dependent resistivity, use a constant resistivity for later comparison:

**4** From the $\alpha$ list, choose **User defined**. Keep the default zero value for $\alpha$.

### *Ground 1*

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

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

**3** From the **Selection** list, choose **connector2**.

### *Electric Potential 1*

1. In the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.
2. In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.
3. In the $V_0$ text field, type `V0`.
4. Locate the **Boundary Selection** section. From the **Selection** list, choose **connector1**.

## MULTIPHYSICS

### *Thermal Expansion 1 (te1)*

1. In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Thermal Expansion 1 (te1)**.
2. In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.
3. Click **Go to Source** for **Volume reference temperature**.

## GLOBAL DEFINITIONS

### *Default Model Inputs*

1. In the **Model Builder** window, under **Global Definitions** click **Default Model Inputs**.
2. In the **Settings** window for **Default Model Inputs**, locate the **Browse Model Inputs** section.
3. Find the **Expression for remaining selection** subsection. In the **Volume reference temperature** text field, type `Text`.

## HEAT TRANSFER IN SOLIDS (HT)

### *Initial Values 1*

1. In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Initial Values 1**.
2. In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
3. In the $T$ text field, type `T0`.

### *Heat Flux 1*

1. In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2. In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **All boundaries**.

   Select all boundaries for simplicity; next you will add a node that overrides this boundary condition for the connectors.
4. Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

**5** In the $h$ text field, type `k`.

**6** In the $T_{ext}$ text field, type `Text`.

*Temperature 1*

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

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

**3** In the $T_0$ text field, type `T0`.

**4** Locate the **Boundary Selection** section. From the **Selection** list, choose **connectors**.

### SOLID MECHANICS (SOLID)

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

*Fixed Constraint 1*

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

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

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

### MESH 1

*Free Tetrahedral 1*

In the **Mesh** toolbar, click **Free Tetrahedral**.

*Size*

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

**2** In the **Settings** window for **Size**, locate the **Element Size** section.

**3** From the **Predefined** list, choose **Finer**.

**4** In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

### STUDY 1

You can use the default solver settings for this model.

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

### RESULTS

*Displacement - Study 1*

The first default plot presents a surface plot of the von Mises stress. Modify it to show the displacement magnitude.

**1** Right-click **Results>Stress (solid)** and choose **Rename**.

**2** In the **Rename 3D Plot Group** dialog box, type `Displacement - Study 1` in the **New label** text field.

**3** Click **OK**.

*Volume 1*

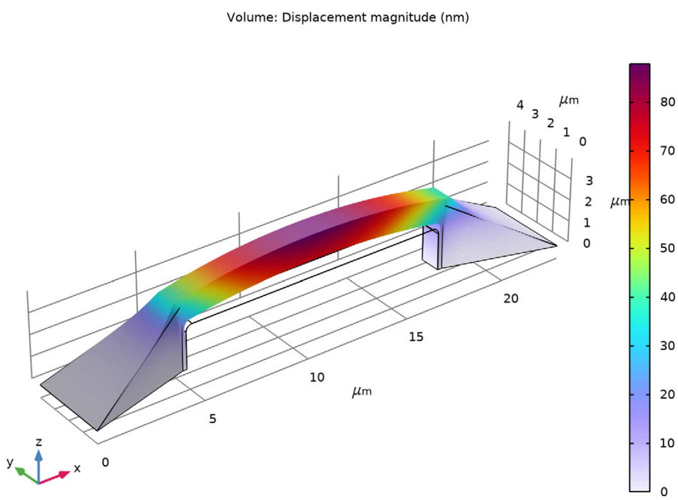
**1** In the **Model Builder** window, expand the **Displacement - Study 1** node, then click **Volume 1**.

**2** In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>solid.disp - Displacement magnitude - m**.

**3** Locate the **Expression** section. From the **Unit** list, choose **nm**.

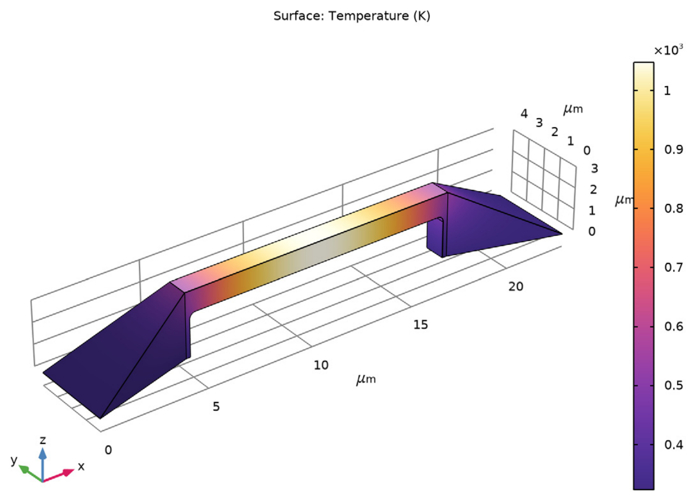
**4** In the **Displacement - Study 1** toolbar, click **Plot**.

As the color legend shows, the maximum displacement is roughly 88 nm with a constant resistivity.



### *Temperature (ht)*

The second default surface plot shows the temperature field. Note the maximum temperature of roughly 1048 K.




Now restore the temperature-dependence of the resistivity that you temporarily disabled and then add a new study and solve the model again.

## ELECTRIC CURRENTS (EC)

### *Current Conservation 1*

1 In the **Model Builder** window, under **Component 1 (comp1)>Electric Currents (ec)** click **Current Conservation 1**.

2 In the **Settings** window for **Current Conservation**, locate the **Constitutive Relation Jc-E** section.

3 From the α list, choose **From material**.

## ADD STUDY

1 In 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 **General Studies>Stationary**.

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

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

#### STUDY 2

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

#### RESULTS

*Temperature (ht) 1*

As you can see from the plot, using the more realistic material model with a temperature-dependent resistivity has a significant effect on the solution. The maximum temperature is now almost 340 K lower.

*Displacement - Study 2*

1 In the **Model Builder** window, right-click **Stress (solid)** and choose **Rename**.
2 In the **Rename 3D Plot Group** dialog box, type `Displacement - Study 2` in the **New label** text field.
3 Click **OK**.

*Volume 1*

1 In the **Model Builder** window, expand the **Displacement - Study 2** node, then click **Volume 1**.
2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>solid.disp - Displacement magnitude - m**.
3 Locate the **Expression** section. From the **Unit** list, choose **nm**.
4 In the **Displacement - Study 2** toolbar, click **Plot**.

   Similarly, the maximum displacement has been reduced from 88 nm to around 50 nm.