Joule Heating of a Microactuator
Introduction

This simple tutorial model simulates the resistive heating—also known as Joule heating—of a two-hot-arm thermal actuator. The model couples the physics phenomena involved in one way only. However, as explained below, you can easily modify it to simulate a two-way coupling between the electric current and the heating of the actuator.

Model Definition

Figure 1 shows the actuator’s parts and dimensions as well as its position on top of a substrate surface.

Figure 1: The thermal microactuator.

Material Data

This model uses the material properties listed in Table 1 for the Joule Heating Model equations. The assumption of constant material properties means that the coupling between physics phenomena is one way only: the electric current through the actuator heats up the material, but the current itself is not affected by the temperature rise. By choosing the linearized resistivity option for the electrical conductivity (available in the
Joule Heating Model node’s settings window and providing the associated material properties you can turn this into a two-way coupling.

**TABLE 1: MATERIAL DATA**

<table>
<thead>
<tr>
<th>PROPERTY</th>
<th>NAME</th>
<th>VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Electrical conductivity</td>
<td>$\sigma$</td>
<td>$5 \cdot 10^4$ S/m</td>
</tr>
<tr>
<td>Relative permeability</td>
<td>$\varepsilon_r$</td>
<td>4.5</td>
</tr>
<tr>
<td>Thermal conductivity</td>
<td>$k$</td>
<td>40 W/(m·K)</td>
</tr>
<tr>
<td>Density</td>
<td>$\rho$</td>
<td>2300 kg/m$^3$</td>
</tr>
<tr>
<td>Heat capacity at constant pressure</td>
<td>$C_p$</td>
<td>600 J/(kg·K)</td>
</tr>
</tbody>
</table>

**BOUNDARY CONDITIONS**

An electric potential is applied between the bases of the hot arms’ anchors. The cold arm anchor and all other surfaces are electrically insulated.

*Figure 2: Electrical boundary conditions.*

The temperature of the base of the three anchors and the three dimples is fixed to that of the substrate’s constant temperature. Because the structure is sandwiched, all other boundaries interact thermally with the surroundings by conduction through thin layers of air.
The heat transfer coefficient is given by the thermal conductivity of air divided by the distance to the surrounding surfaces for the system. This exercise uses different heat transfer coefficients for the actuator’s upper and other surfaces.

\[
\text{Heat flux}_\perp = h(T - T_{\text{amb}})
\]

*Figure 3: Heat-transfer boundary conditions.*

**Results**

*Figure 4* shows the temperature distribution on the actuator’s surface. The line graph in *Figure 5* provides more detailed information about the temperature along a single edge facing the substrate plane.
Figure 4: The temperature distribution on the actuator surface.

Figure 5: Temperature along the actuators longest edge facing the substrate.
Application Library path: COMSOL_Multiphysics/Multiphysics/thermal_actuator_jh

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 Heat Transfer>Electromagnetic Heating>Joule Heating.
3 Click Add.
4 Click Study.
5 In the Select Study tree, select Preset Studies for Selected Physics Interfaces>Stationary.
6 Click Done.

COMPONENT 1 (COMP1)
1 In the Model Builder window, right-click Component 1 (comp1) and choose Rename.
2 In the Rename Component dialog box, type Thermal Actuator in the New label text field.
3 Click OK.

GLOBAL DEFINITIONS

Parameters
1 On the Home toolbar, click Parameters.
2 In the Settings window for Parameters, locate the Parameters section.
In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Name</th>
<th>Expression</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>htc_s</td>
<td>$0.04 \text{W/(m}^*\text{K)} / 2[\text{um}]$</td>
<td>20000 $\text{W/(m}^2\text{·K)}$</td>
<td>Heat transfer coefficient</td>
</tr>
<tr>
<td>htc_us</td>
<td>$0.04 \text{W/(m}^*\text{K)} / 100[\text{um}]$</td>
<td>400 $\text{W/(m}^2\text{·K)}$</td>
<td>Heat transfer coefficient, upper surface</td>
</tr>
<tr>
<td>DV</td>
<td>5$\text{[V]}$</td>
<td>5 V</td>
<td>Applied voltage</td>
</tr>
</tbody>
</table>

**GEOMETRY 1**

*Import 1 (imp1)*

1. On the **Home** toolbar, click **Import**.
2. In the **Settings** window for Import, locate the **Import** section.
3. Click **Browse**.
4. Browse to the application’s Application Libraries folder and double-click the file thermal_actuator.mphbin.
5. Click **Build All Objects**.
6. Click the **Zoom Extents** button on the **Graphics** toolbar.

**DEFINITIONS**

*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 **Substrate Contact** 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 10, 30, 50, 70, 76, and 82 only.

**MATERIALS**

In the **Model Builder** window, under **Thermal Actuator (comp1)** right-click **Materials** and choose **Blank Material**.

*Material 1 (mat1)*

By default, the first material you define applies to all domains.
1. In the **Settings** window for Material, locate the **Material Contents** section.

2. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Property</th>
<th>Name</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Electrical conductivity</td>
<td>sigma</td>
<td>5e4</td>
<td>S/m</td>
<td>Basic</td>
</tr>
<tr>
<td>Relative permittivity</td>
<td>epsilonr</td>
<td>4.5</td>
<td>F/m</td>
<td>Basic</td>
</tr>
<tr>
<td>Thermal conductivity</td>
<td>k</td>
<td>40</td>
<td>W/(m·K)</td>
<td>Basic</td>
</tr>
<tr>
<td>Density</td>
<td>rho</td>
<td>2.3e3</td>
<td>kg/m³</td>
<td>Basic</td>
</tr>
<tr>
<td>Heat capacity at constant pressure</td>
<td>Cp</td>
<td>600</td>
<td>J/(kg·K)</td>
<td>Basic</td>
</tr>
</tbody>
</table>

**ELECTRIC CURRENTS (EC)**

*Ground 1*

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

2. Select Boundary 10 only.

**Electric Potential 1**

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

2. Select Boundary 30 only.

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

4. In the $V_0$ text field, type $DV$.

**HEAT TRANSFER IN SOLIDS (HT)**

In the **Model Builder** window, under **Thermal Actuator (comp1)** click **Heat Transfer in Solids (ht)**.

**Heat Flux 1**

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

   This boundary condition applies to all boundaries except the top-surface boundary and those in contact with the substrate. A Temperature condition on the **substrate_contact** boundaries will override this Heat Flux condition so you do not explicitly need to exclude those boundaries. In contrast, because the Heat Flux boundary condition is additive, you must explicitly exclude the top-surface boundary from the selection. Implement this selection as follows:

2. In the **Settings** window for Heat Flux, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All boundaries**. In the **Graphics** window, click on the top surface and then right-click to remove it from the selection.

A convective heat flux is used to model the heat flux through a thin air layer. The heat transfer coefficient, $htc_s$, is defined as the ratio of the air thermal conductivity to the gap thickness.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

5 In the $h$ text field, type $htc_s$.

**Heat Flux 2**

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

2 Select Boundary 4 only.

A convective heat flux is used to model the heat flux through a thin air layer. The heat transfer coefficient, $htc_us$, is defined as the ratio of the air thermal conductivity to the gap thickness.

3 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.

4 Click the **Convective heat flux** button.

5 In the $h$ text field, type $htc_us$.

**Temperature 1**

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

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

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

**MESH 1**

1 In the **Model Builder** window, under **Thermal Actuator (comp1)** click **Mesh 1**.

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

3 From the **Element size** list, choose **Fine**.

**Free Triangular 1**

1 Right-click **Thermal Actuator (comp1)>Mesh 1** and choose **More Operations>Free Triangular**.

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type 3 in the **Selection** text field.

6 Click **OK**.
7 In the Settings window for Free Triangular, click Build Selected.

Swept 1
1 In the Model Builder window, right-click Mesh 1 and choose Swept.
2 In the Model Builder window, click Mesh 1.
3 In the Settings window for Mesh, click Build All.

STUDY 1
On the Home toolbar, click Compute.

RESULTS

Electric Potential (ec)
The first default plot group shows the electric potential distribution.

Temperature (ht)
The second default plot group shows the temperature distribution on the surface (see Figure 4).

1 Click the Zoom Extents button on the Graphics toolbar.

Reproduce the plot in Figure 5 by following these steps:
1D Plot Group 4
On the Home toolbar, click Add Plot Group and choose 1D Plot Group.

Line Graph 1
On the 1D Plot Group 4 toolbar, click Line Graph.

1D Plot Group 4
1 Select Edge 52 only.
2 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 Model>Thermal Actuator>Heat Transfer in Solids>Temperature>T - Temperature.
3 Click Replace Expression in the upper-right corner of the x-axis data section. From the menu, choose Model>Thermal Actuator>Geometry>Coordinate>x - x-coordinate.
4 Locate the x-Axis Data section. From the Unit list, choose µm.
5 On the 1D Plot Group 4 toolbar, click Plot.