Created in COMSOL Multiphysics 5.6

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.

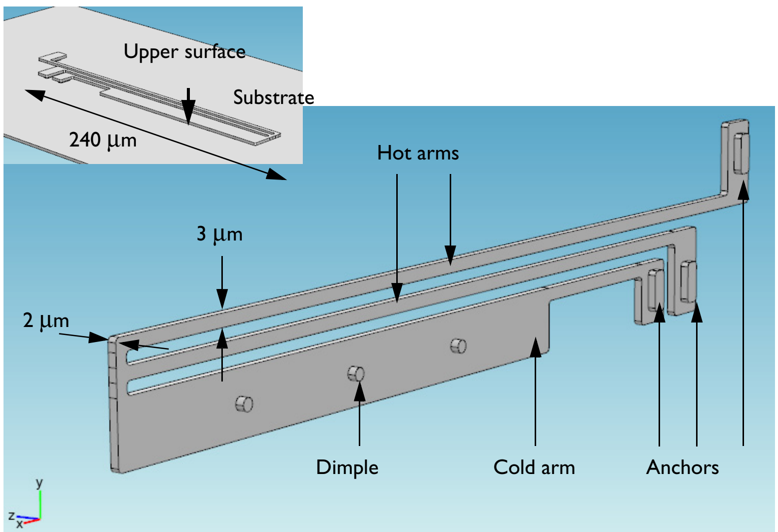


Figure 1: The thermal microactuator.

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 using a material where the electrical conductivity is temperature dependent, you can turn this into a two-way coupling.

TABLE 1: MATERIAL DATA.

PROPERTY	NAME	VALUE
Electrical conductivity	σ	$5\cdot10^4$ S/m
Relative permeability	ε_r	4.5
Thermal conductivity	k	40 W/(m·K)
Density	ρ	2300 kg/m^3
Heat capacity at constant pressure	C_p	600 J/(kg·K)

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.

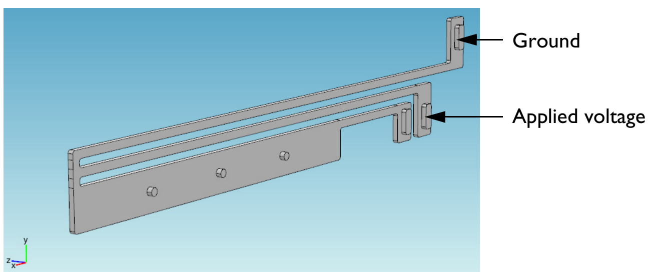


Figure 2: Electrical boundary conditions.

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.

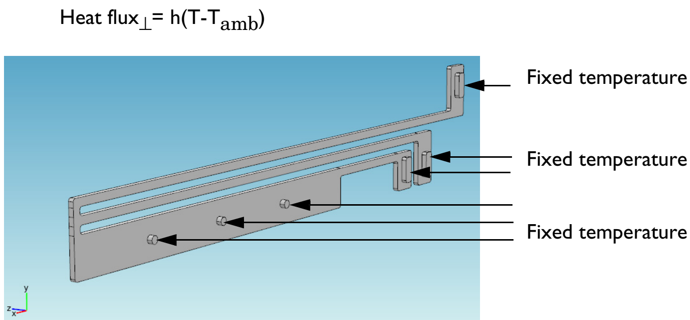


Figure 3: Heat-transfer boundary conditions.

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.

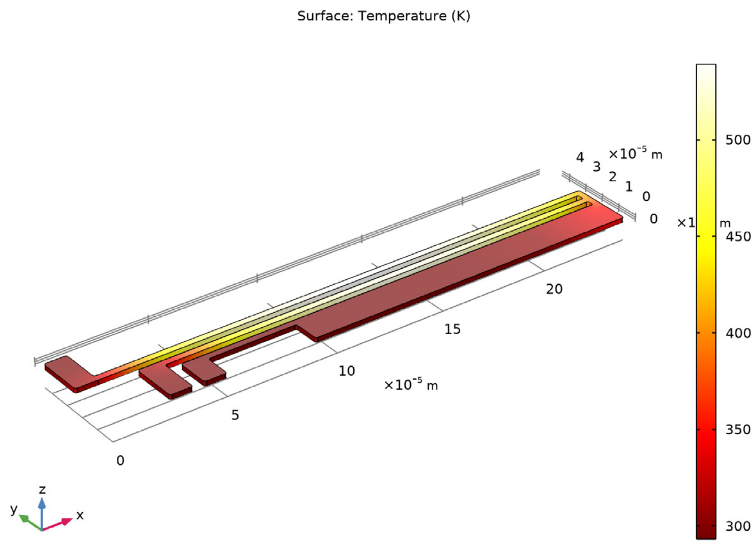


Figure 4: The temperature distribution on the actuator surface.

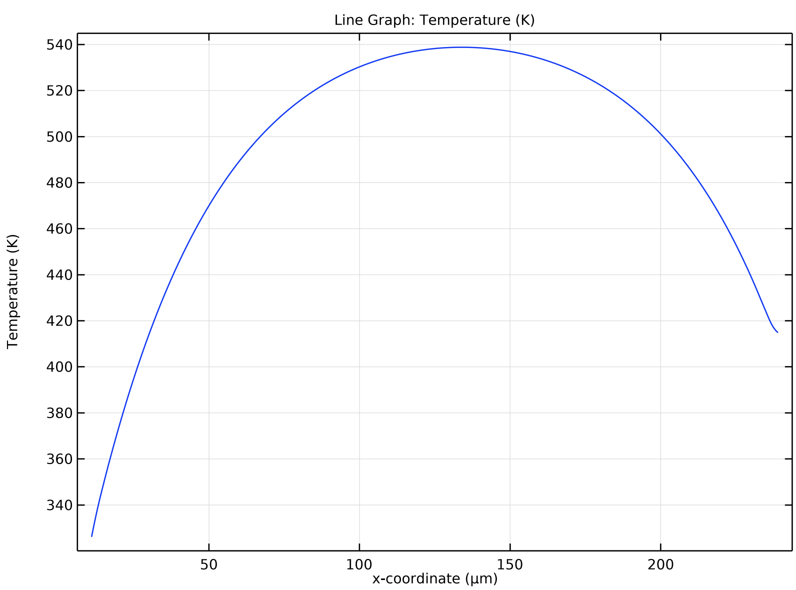


Figure 5: Temperature along the actuators longest edge facing the substrate.

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 **General Studies>Stationary**.
6 Click **Done**.

THERMAL ACTUATOR

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

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
htc_s	0.04[W/(m*K)]/2[um]	20000 W/(m²·K)	Heat transfer coefficient
htc_us	0.04[W/(m*K)]/100[um]	400 W/(m²·K)	Heat transfer coefficient, upper surface
DV	5[V]	5 V	Applied voltage

GEOMETRY 1

Import 1 (imp1)

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

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

3 Click **Browse**.

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

5 Click **Build All Objects**.

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

DEFINITIONS

Substrate Contact

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

2 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

Material 1 (mat1)

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

By default, the first material you define applies to all domains.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	5e4	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	4.5	1	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	40	W/(m·K)	Basic
Density	rho	2.3e3	kg/m³	Basic
Heat capacity at constant pressure	Cp	600	J/(kg·K)	Basic

ELECTRIC CURRENTS (EC)

Ground 1

1 In the **Model Builder** window, under **Thermal Actuator (comp1)** right-click **Electric Currents (ec)** and choose **Ground**.

2 Select Boundary 10 only.

Electric Potential 1

1 In 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 In 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**.

4 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.

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

6 In the h text field, type `htc_s`.

Heat Flux 2

1 In 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 In 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 **Physics-Controlled Mesh** section.

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

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **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.

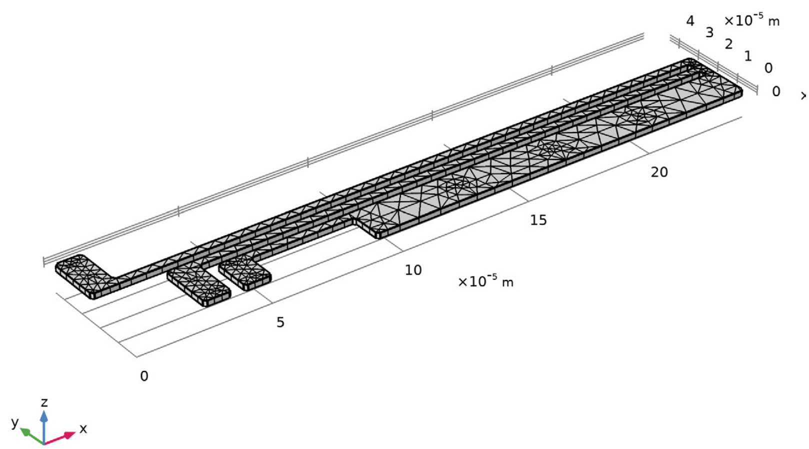
6 Click **OK**.

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

Swept 1

1 In the **Mesh** toolbar, click **Swept**.

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



STUDY 1

In 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 in the **Graphics** toolbar.

Reproduce the plot in Figure 5 by following these steps:

1D Plot Group 4

In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

Line Graph 1

1 Right-click **1D Plot Group 4** and choose **Line Graph**.
2 Select Edge 52 only.
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 **Thermal Actuator (comp1)> Heat Transfer in Solids>Temperature>T - Temperature - K**.
4 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Thermal Actuator (comp1)>Geometry>Coordinate>x - x-coordinate**.
5 Locate the **x-Axis Data** section. From the **Unit** list, choose **µm**.
6 In the **1D Plot Group 4** toolbar, click **Plot**.