



Model created in COMSOL Multiphysics 6.4

Thermal Actuator

Introduction

For a description of this model, see [Thermal Actuator — Parameterized](#), which describes a version of the same model (called `thermal_actuator_tem_parameterized`) that only differs in the way the geometry is created; while the modeling instructions below describe how you can import the finished geometry from an MPHBIN-file, the instructions in the above referenced model detail the steps required to create the geometry in the COMSOL Desktop.

Reference


1. D.M. Burns and V.M. Bright, “Design and performance of a double hot arm polysilicon thermal actuator,” *Proc. SPIE 3224, Micromachined Devices and Components III*, 1997; doi: [10.1117/12.284528](https://doi.org/10.1117/12.284528).

Application Library path: Structural_Mechanics_Module/Thermal-Structure_Interaction/thermal_actuator_tem




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**.

THERMAL ACTUATOR

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

- 2 In the **Rename Component** dialog, 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	$0.04 [W / (m \cdot K)] / 2 [\mu m]$	20000 W/(m ² ·K)	Heat transfer coefficient
htc_us	$0.04 [W / (m \cdot K)] / 100 [\mu m]$	400 W/(m ² ·K)	Heat transfer coefficient, upper surface
DV	5[V]	5 V	Applied voltage


GEOMETRY 1

Import 1 (impl)



- 1 In the **Geometry** toolbar, click  **Import**.
- 2 In the **Settings** window for **Import**, locate the **Source** 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  **Go to Default View** button in the **Graphics** toolbar.

DEFINITIONS

substrate contact

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 10, 30, 50, 70, 76, and 82 only.
- 5 In the **Label** text field, type substrate contact.

ADD MATERIAL

- 1 In the **Materials** toolbar, click  **Add Material** to open the **Add Material** window.
- 2 Go to the **Add Material** window.
- 3 In the tree, select **MEMS > Semiconductors > Si - Polycrystalline silicon**.
- 4 Click the **Add to Component** button in the window toolbar.
- 5 In the **Materials** toolbar, click  **Add Material** to close the **Add Material** window.

MATERIALS

Si - Polycrystalline silicon (mat1)


By default, the first material you add applies on all domains so you can keep the **Geometric Entity Selection** settings.

- 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
Electric conductivity	sigma_iso ; sigma_ii = sigma_iso, sigma_ij = 0	5e4	S/m	Basic

SOLID MECHANICS (SOLID)

Fixed Constraint 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Fixed Constraint**.
- 2 Select Boundaries 10, 30, and 50 only.

Roller 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Roller**.
- 2 Select Boundaries 70, 76, and 82 only.

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 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, h_{tc_s} is defined as the ratio of the air thermal conductivity to the gap thickness.
- 5 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.
- 6 In the h text field, type h_{tc_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, h_{tc_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 From the **Flux type** list, choose **Convective heat flux**.
- 5 In the h text field, type h_{tc_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**.

ELECTRIC CURRENTS (EC)

Ground 1


- 1 In the **Physics** toolbar, click  **Boundaries** 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.

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 **Extra fine**.
- 4 Click  **Build All**.


STUDY 1


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 **Study Settings** section.
- 3 Select the **Include geometric nonlinearity** checkbox.

The default solver sequence is not optimal for this problem. The only deviation from linearity comes from the geometric nonlinearity. Both the **Electric Currents** and **Temperature** groups in the segregated solver contain damping intended to make solution of nonlinear problems more stable, but slower. Removing that damping gives a significant speed up. Also, if iterations are performed for the **Solid Mechanics** group where the actual nonlinearity is present, there will be fewer iterations over the whole set of segregated groups.

Solution 1 (sol1)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
- 3 In the **Model Builder** window, expand the **Study 1 > Solver Configurations > Solution 1 (sol1) > Stationary Solver 1 > Segregated 1** node, then click **Electric Currents**.
- 4 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.
- 5 In the **Damping factor** text field, type 1.
- 6 In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1) > Stationary Solver 1 > Segregated 1** click **Temperature**.
- 7 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.
- 8 In the **Damping factor** text field, type 1.
- 9 In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1) > Stationary Solver 1 > Segregated 1** click **Solid Mechanics**.



- 10 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.
- 11 From the **Termination technique** list, choose **Tolerance**.
- 12 Locate the **General** section. From the **Linear solver** list, choose **Suggested Iterative Solver solid (te1)**.
- 13 In the **Study** toolbar, click  **Compute**.

RESULTS


Stress (solid)

The first default plot show the von Mises stress.

Volume 1


- 1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Volume 1**.
- 2 In the **Settings** window for **Volume**, locate the **Expression** section.
- 3 From the **Unit** list, choose **MPa**.
- 4 In the **Stress (solid)** toolbar, click  **Plot**.
- 5 Click the  **Zoom Extents** button in the **Graphics** toolbar.

Temperature (ht)

- 1 Click the  **Go to Default View** button in the **Graphics** toolbar.
- The second default plot shows the temperature field.

Create a new plot for displacement.


Displacement

- 1 In the **Results** toolbar, click  **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type Displacement in the **Label** text field.

Surface 1

- 1 Right-click **Displacement** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **µm**.
- 4 Locate the **Coloring and Style** section. From the **Color table** list, choose **SpectrumLight**.

Deformation 1

- 1 Right-click **Surface 1** and choose **Deformation**.
- 2 In the **Displacement** toolbar, click  **Plot**.

