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.

Application Library path: `MEMS_Module/Actuators/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*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 **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 ; sigmaii = sigma_iso, sigmaij = 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, `htc_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 `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. From the **Flux type** list, choose **Convective heat flux**.
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**.

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**.