Created in COMSOL Multiphysics 6.0

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: `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 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 **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 Right-click **Explicit 1** and choose **Rename**.

6 In the **Rename Explicit** dialog box, type `substrate contact` in the **New label** text field.

7 Click **OK**.

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>Semiconductors>Si - Polycrystalline silicon**.

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

5 In the **Home** 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
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	5e4	S/m	Basic

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 In the **Model Builder** window, under **Thermal Actuator (comp1)** right-click **Solid Mechanics (solid)** 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)

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 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)

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

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 **Fine**.

Size 1

1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.

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

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

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundaries 86–91 only.

6 In the **Model Builder** window, right-click **Mesh 1** and choose **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** check box.

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

The first default plot show the von Mises stress.

RESULTS

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 **Home** toolbar, click **Add Plot Group** and choose **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**.