

Thermal Actuator — Parameterized

Introduction

This example model consists of a two-hot-arm thermal actuator made of polysilicon. The actuator is activated through thermal expansion. The temperature increase required to deform the two hot arms, and thus displace the actuator, is obtained through Joule heating (resistive heating). The greater expansion of the hot arms, compared to the cold arm, causes a bending of the actuator. In an actual device, a wide range of electrical resistance values is possible through doping of the polysilicon material.

The actuator's operation thus involves three coupled physics phenomena: electric current conduction, heat conduction with heat generation, and structural stresses and strains due to thermal expansion.

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.

BOUNDARY CONDITIONS AND CONSTRAINTS

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. This can be implemented as thermal contact conditions or as a convective heat flux condition, where the heat flux coefficient represent one over the thermal resistance. In this model we chose to use a heat flux condition. 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.

All three arms are mechanically fixed at the base of the three anchors. The dimples can move freely in the plane of the substrate (the *xy*-plane in the figure) but do not move in the direction perpendicular to the substrate (the *z* direction).



Figure 4: Structural boundary conditions and constraints.

Results

Figure 5 shows the surface temperature distribution for the actuator. Figure 6 illustrates the displacement field through color and deformation plot.



Figure 5: Temperature plot.

Surface: Displacement magnitude (µm)



Figure 6: Displacement in the thermal actuator.

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_parameterized

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click 🙆 Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 间 3D.
- 2 In the Select Physics tree, select Structural Mechanics>Thermal-Structure Interaction> Joule Heating and Thermal Expansion.
- 3 Click Add.
- 4 Click \bigcirc Study.
- 5 In the Select Study tree, select General Studies>Stationary.
- 6 Click **M** Done.

THERMAL ACTUATOR

- I In the Model Builder window, right-click Component I (compl) 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

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
d	3[um]	3E-6 m	Height of the hot arm
dw	15[um]	1.5E-5 m	Height of the cold arm
gap	3[um]	3E-6 m	Gap between arms
wb	10[um]	IE-5 m	Width of the base
WV	25[um]	2.5E-5 m	Difference in length between hot arms
L	240[um]	2.4E-4 m	Actuator length
L1	L-wb	2.3E-4 m	Length of the longest hot arm
L2	L-wb-wv	2.05E-4 m	Length of the shortest hot arm

Name Expression		Value	Description		
L3	L-2*wb-wv-L/48-L/6	1.5E-4 m	Length of the cold arm, thick part		
L4	L/6	4E-5 m	Length of the cold arm, thin part		
htc_s	0.04[W/(m*K)]/ 2[um]	20000 W/(m ² ·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 I

Work Plane I (wp1)

I In the Geometry toolbar, click 😓 Work Plane.

2 In the Settings window for Work Plane, click 틤 Build Selected.

3 Click 📥 Show Work Plane.

Work Plane I (wp1)>Rectangle I (r1)

- I In the Work Plane toolbar, click Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type L3.
- 4 In the **Height** text field, type dw.
- 5 Locate the Position section. In the xw text field, type L-L3.
- 6 Click 틤 Build Selected.

Work Plane 1 (wp1)>Rectangle 2 (r2)

- I In the Work Plane toolbar, click Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type L4.
- **4** In the **Height** text field, type d.
- 5 Locate the Position section. In the xw text field, type L-L3-L4.
- 6 In the **yw** text field, type dw-d.
- 7 Click 📄 Build Selected.
- 8 Click the 🕂 Zoom Extents button in the Graphics toolbar.

Work Plane 1 (wp1)>Rectangle 3 (r3)

- I In the Work Plane toolbar, click Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type wb.
- 4 In the **Height** text field, type dw.
- 5 Locate the Position section. In the xw text field, type L-L3-L4-wb.
- 6 Click 틤 Build Selected.
- 7 Click the **Zoom Extents** button in the **Graphics** toolbar.

Work Plane I (wp1)>Rectangle 4 (r4)

- I In the Work Plane toolbar, click Mectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type L2.
- 4 In the **Height** text field, type d.
- 5 Locate the **Position** section. In the **xw** text field, type L-L2.
- 6 In the **yw** text field, type dw+gap.
- 7 Click 틤 Build Selected.

Work Plane 1 (wp1)>Rectangle 5 (r5)

- I In the Work Plane toolbar, click 📃 Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type wb.
- **4** In the **Height** text field, type dw+gap+d.
- 5 Locate the Position section. In the xw text field, type L-L2-wb.
- 6 Click 틤 Build Selected.
- 7 Click the **Zoom Extents** button in the **Graphics** toolbar.

Work Plane 1 (wp1)>Rectangle 6 (r6)

- I In the Work Plane toolbar, click 📃 Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type L1.
- 4 In the **Height** text field, type d.
- 5 Locate the Position section. In the xw text field, type L-L1.
- 6 In the **yw** text field, type dw+d+2*gap.

- 7 Click 틤 Build Selected.
- 8 Click the \longleftrightarrow Zoom Extents button in the Graphics toolbar.

Work Plane 1 (wp1)>Rectangle 7 (r7)

- I In the Work Plane toolbar, click 📃 Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type wb.
- 4 In the **Height** text field, type dw+gap+d.
- 5 Locate the Position section. In the yw text field, type dw+d+2*gap.
- 6 Click 틤 Build Selected.
- 7 Click the 4 Zoom Extents button in the Graphics toolbar.

Work Plane 1 (wp1)>Rectangle 8 (r8)

- I In the Work Plane toolbar, click 📃 Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type d.
- **4** In the **Height** text field, type gap.
- 5 Locate the Position section. In the xw text field, type L-d.
- **6** In the **yw** text field, type dw+gap+d.
- 7 Click 틤 Build Selected.

Work Plane 1 (wp1)>Rectangle 9 (r9)

- I In the Work Plane toolbar, click Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type d.
- **4** In the **Height** text field, type gap.
- 5 Locate the Position section. In the xw text field, type L-d.
- 6 In the **yw** text field, type dw.
- 7 Click 틤 Build Selected.

Work Plane I (wp1)>Union I (uni1)

- I In the Work Plane toolbar, click i Booleans and Partitions and choose Union.
- 2 Click in the Graphics window and then press Ctrl+A to select all objects.
- 3 In the Settings window for Union, locate the Union section.
- 4 Clear the Keep interior boundaries check box.

5 Click 📄 Build Selected.

Work Plane I (wp1)>Fillet I (fill)

- I In the Work Plane toolbar, click 🥢 Fillet.
- 2 In the Settings window for Fillet, locate the Radius section.
- 3 In the Radius text field, type d/3.
- 4 On the object unil, select Points 1, 2, 4–9, 11–14, 16, 17, 19–23, and 28 only.

It might be easier to select the points by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)



5 Click 틤 Build Selected.

Extrude I (extI)

- In the Model Builder window, under Thermal Actuator (compl)>Geometry | right-click
 Work Plane | (wpl) and choose Extrude.
- 2 In the Settings window for Extrude, locate the Distances section.
- **3** In the table, enter the following settings:

Distances (m)

2e-6

- 4 Click 틤 Build Selected.
- **5** Click the **J Go to Default View** button in the **Graphics** toolbar.

Work Plane 2 (wp2)

- I In the Geometry toolbar, click 📥 Work Plane.
- 2 In the Settings window for Work Plane, click 틤 Build Selected.
- 3 Click 📥 Show Work Plane.

Work Plane 2 (wp2)>Plane Geometry

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

Work Plane 2 (wp2)>Rectangle 1 (r1)

- I In the Work Plane toolbar, click Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type wb-2*d.
- 4 In the **Height** text field, type 2.5*(wb-2*d).
- 5 Locate the **Position** section. In the **xw** text field, type d.
- 6 In the yw text field, type (dw+d+2*gap)+(dw+gap+d)-2.5*(wb-2*d)-d.
- 7 Click 📄 Build Selected.

Work Plane 2 (wp2)>Rectangle 2 (r2)

- I In the Work Plane toolbar, click Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type wb-2*d.
- 4 In the **Height** text field, type 2.5*(wb-2*d).
- 5 Locate the Position section. In the xw text field, type L-L2-wb+d.
- **6** In the **yw** text field, type **d**.
- 7 Click 틤 Build Selected.

Work Plane 2 (wp2)>Rectangle 3 (r3)

- I In the Work Plane toolbar, click Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type wb-2*d.
- 4 In the **Height** text field, type 2.5*(wb-2*d).
- 5 Locate the Position section. In the xw text field, type L-L3-L4-wb+d.
- **6** In the **yw** text field, type **d**.

7 Click 틤 Build Selected.

Work Plane 2 (wp2)>Fillet 1 (fill)

- I In the Work Plane toolbar, click 🥢 Fillet.
- 2 In the Settings window for Fillet, locate the Radius section.
- 3 In the Radius text field, type d/3.
- **4** Select all four vertices for all three rectangles as follows:
- **5** Click the **Select Box** button in the **Graphics** toolbar.
- **6** In the **Graphics** window, draw a box encompassing the three rectangles you just created, then right-click to confirm the selection.



7 Click 틤 Build Selected.

Work Plane 2 (wp2)>Circle 1 (c1)

- I In the Work Plane toolbar, click 🕑 Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type d/2.
- 4 Locate the Position section. In the xw text field, type L-L3/4.
- **5** In the **yw** text field, type dw/2.
- 6 Click 📄 Build Selected.

Work Plane 2 (wp2)>Circle 2 (c2)

- I In the Work Plane toolbar, click (•) Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type d/2.
- 4 Locate the Position section. In the xw text field, type L-L3/2.
- **5** In the **yw** text field, type dw/2.
- 6 Click 틤 Build Selected.

Work Plane 2 (wp2)>Circle 3 (c3)

- I In the Work Plane toolbar, click 😶 Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type d/2.
- 4 Locate the Position section. In the xw text field, type L-3*L3/4.
- **5** In the **yw** text field, type dw/2.
- 6 Click 📄 Build Selected.

Extrude 2 (ext2)

- In the Model Builder window, under Thermal Actuator (compl)>Geometry | right-click
 Work Plane 2 (wp2) and choose Extrude.
- 2 In the Settings window for Extrude, locate the Distances section.
- **3** In the table, enter the following settings:

Distances (m)

2e-6

- 4 Select the **Reverse direction** check box.
- 5 Click 틤 Build Selected.

Union I (uniI)

- I In the Geometry toolbar, click 🔲 Booleans and Partitions and choose Union.
- **2** Click the 4 **Zoom Extents** button in the **Graphics** toolbar.
- 3 Click in the Graphics window and then press Ctrl+A to select both objects.
- **4** In the Settings window for Union, click 📗 Build All Objects.

DEFINITIONS

substrate contact

- I In the Definitions toolbar, click http://www.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 Click the Solution of the Graphics toolbar three times to view the geometry from below.



- 6 In the Model Builder window, right-click Explicit I and choose Rename.
- 7 In the Rename Explicit dialog box, type substrate contact in the New label text field.
- 8 Click OK.

ADD MATERIAL

- I 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.

I 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 I

- I In the Model Builder window, under Thermal Actuator (compl) right-click Solid Mechanics (solid) and choose Fixed Constraint.
- 2 Select Boundaries 10, 30, and 50 only.

Roller I

- I 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 (compl) click Heat Transfer in Solids (ht).

Heat Flux 1

I 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. From the Flux type list, choose Convective heat flux.
- 6 In the *h* text field, type htc_s.

Heat Flux 2

- I 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

- I 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 (compl) click Electric Currents (ec).

Ground I

- I In the Physics toolbar, click 🔚 Boundaries and choose Ground.
- **2** Select Boundary 10 only.

Electric Potential 1

- I 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 I

Free Tetrahedral I

In the Mesh toolbar, click \bigwedge Free Tetrahedral.

Size I

Right-click Free Tetrahedral I and choose Size.

Size

- I In the Settings window for Size, locate the Element Size section.
- 2 From the **Predefined** list, choose **Fine**.
- **3** Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section. In the **Maximum element growth rate** text field, type 1.2.

This setting makes the mesh more robust for parametric sweeps over the geometry length parameter L.

Size 1

- I In the Model Builder window, under Thermal Actuator (compl)>Mesh l> Free Tetrahedral I click Size I.
- 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 Click 📗 Build All.

STUDY I

Step 1: Stationary

- I In the Model Builder window, under Study I click Step I: 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

- I In the Model Builder window, expand the Stress (solid) node, then click Volume I.
- 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 **I** Plot.
- **5** Click the 4 **Zoom Extents** button in the **Graphics** toolbar.

Temperature (ht)

I Click the $\sqrt{-}$ Go to Default View button in the Graphics toolbar.

The second default plot shows the temperature field.

Create a new plot for displacement.

Displacement

- I 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

- I 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

- I Right-click Surface I and choose Deformation.
- 2 In the Displacement toolbar, click **Displacement** 100 Plot.