Created in COMSOL Multiphysics 5.6

Thermal Microactuator Simplified

This model is licensed under the COMSOL Software License Agreement 5.6.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

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.

The material properties of polysilicon are temperature dependent, which means that the involved physics phenomena are fully coupled. The electric current through the hot arms increases the temperature in the actuator, which in turn causes thermal expansion and changes the electrical conductivity of the 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.

In this example the thermal expansion is included manually using the Equation View. Furthermore, only linear strains are considered, which is a valid approximation provided deformations are small. Using the Structural Mechanics Module or the MEMS Module, you can directly include the thermal equation in the physics and also take into account possible large deformations; see the model Thermal Microactuator in the Structural Mechanics Module or MEMS Module Application Libraries.

Model Definition

Figure 1 shows the actuator's parts and dimensions as well as its position on top of a substrate surface.

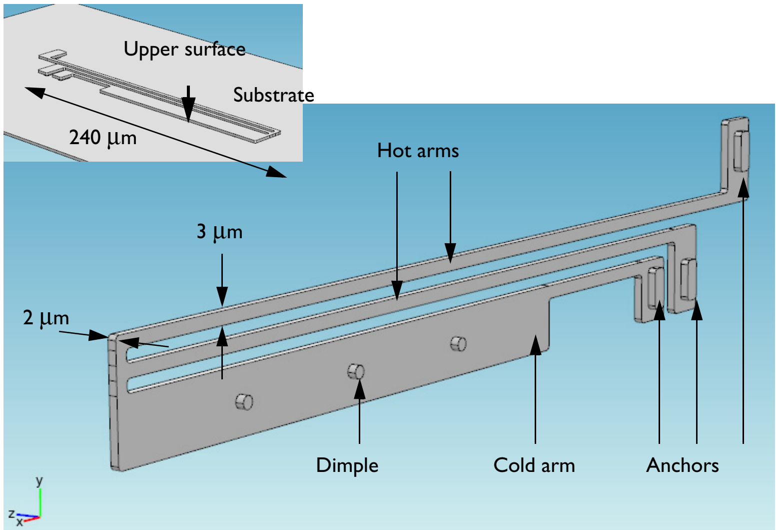


Figure 1: The thermal microactuator.

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.

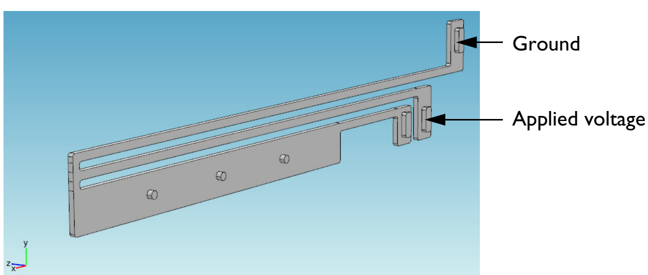


Figure 2: Electrical boundary conditions.

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.

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.

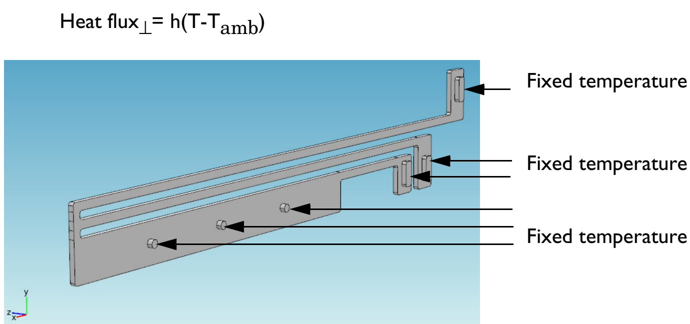


Figure 3: Heat-transfer boundary conditions.

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).

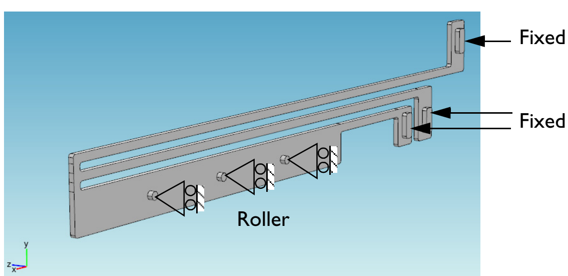


Figure 4: Structural boundary conditions and constraints.

GEOMETRY PROGRAMMING OPERATIONS

The model geometry uses a conditional If + EndIf statement to implement two distinct designs: a two-arms or a three-arms actuator design. It is possible to switch from one design to the other by means of a parameter.

Results

Figure 5 shows the surface temperature distribution for the actuator design with two arms. It also illustrates the displacement field through a deformation plot.

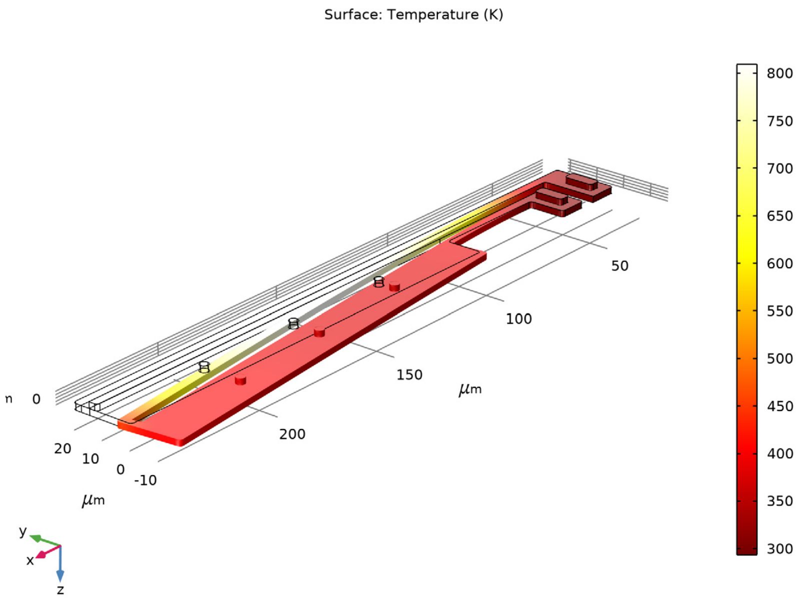


Figure 5: Temperature (surface) and displacement (deformation) in the two-arms actuator design.

For the two-arms design, the computed displacement at the tip of the actuator is about 3.3 μm.

Figure 6 shows the surface temperature distribution for the actuator design with 3 arms only.

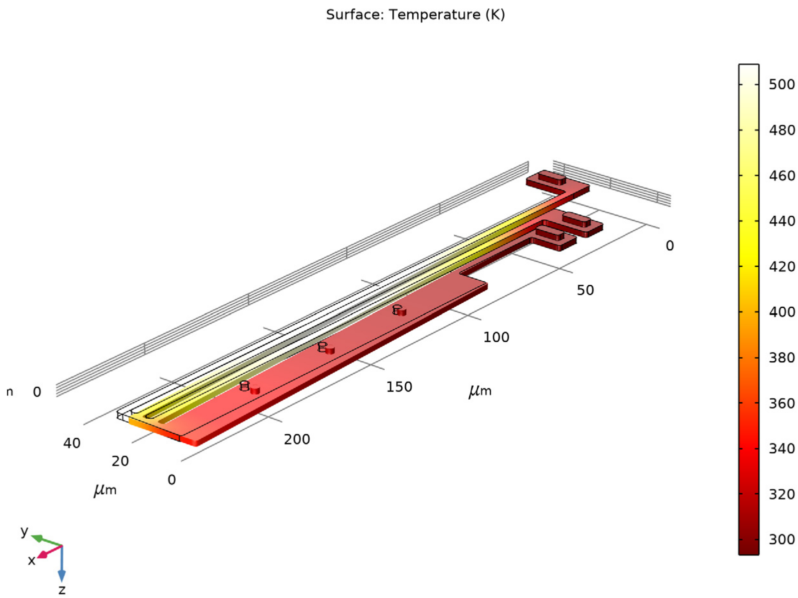


Figure 6: Temperature (surface) and displacement (deformation) in the three arms actuator design.

For the two-arms design, the computed displacement at the tip of the actuator is about 1.1 µm.

Notes About the COMSOL Implementation

Using Equation View, you can access the equation level. There you can modify the strain definition used by the linear elastic material model.

The thermal strain to be added is:

$$\varepsilon_{th} = \alpha \cdot (T - T_0)$$

Here α is the material thermal expansion coefficient, T the current temperature, and T_0 the strain reference temperature.

To enable the equation view, click the **Show** button ( ) and choose **Equation View**. You can then access the equation view by expanding each node of the model tree.

Application Library path: `COMSOL_Multiphysics/Multiphysics/thermal_actuator_simplified`

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 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Stationary**.

8 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**
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]	1E-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
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/(m^2·K)	Heat transfer coefficient, upper surface
DV	5[V]	5 V	Applied voltage
alphaps	2.6e-6[1/K]	2.6E-6 1/K	Coefficient of thermal expansion
T0	293.15[K]	293.15 K	Strain reference temperature
noa	3	3	Number of arms

GEOMETRY 1

1 In the **Model Builder** window, under **Thermal Actuator (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **µm**.

If 1 (if1)

1 In the **Geometry** toolbar, click **Programming** and choose **If + End If**.

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

3 In the **Condition** text field, type (noa==3).

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Rectangle 1 (r1)

1 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. Leave **yw** at the default value 0.

Rectangles 2-9

Proceed to create eight additional rectangles with the following settings:

Name	Width	Height	xw	yw
Rectangle 2 (r2)	L4	d	L-L3-L4	dw-d
Rectangle 3 (r3)	wb	dw	L-L3-L4-wb	0
Rectangle 4 (r4)	L2	d	L-L2	dw+gap
Rectangle 5 (r5)	wb	dw+gap+d	L-L2-wb	0
Rectangle 6 (r6)	L1	d	L-L1	dw+d+2*gap
Rectangle 7 (r7)	wb	dw+gap+d	0	dw+d+2*gap
Rectangle 8 (r8)	d	gap	L-d	dw+gap+d
Rectangle 9 (r9)	d	gap	L-d	dw

Work Plane 1 (wp1)>Union 1 (uni1)

1 In the **Work Plane** toolbar, click **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.

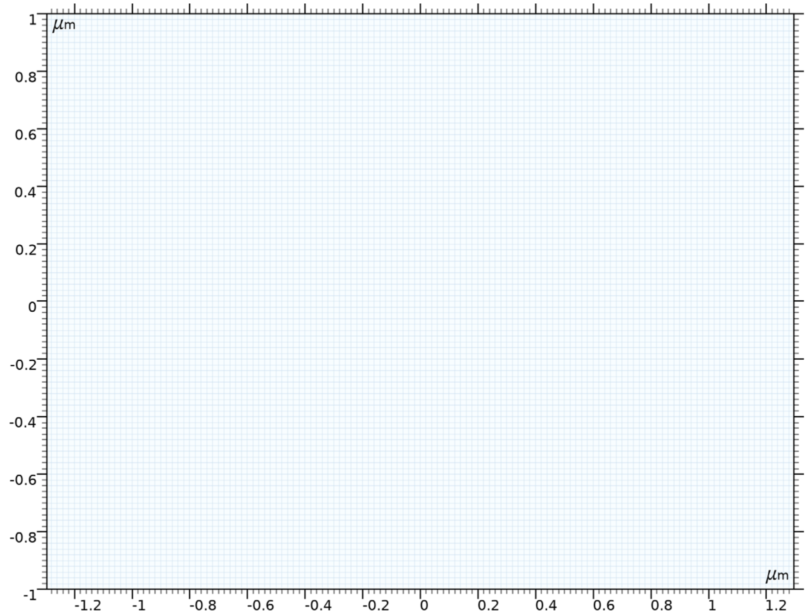
4 Clear the **Keep interior boundaries** check box.

Work Plane 1 (wp1)>Fillet 1 (fil1)

1 In the **Work Plane** toolbar, click **Fillet**.

2 On the object **uni1**, 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.)



3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `d/3`.

Extrude 1 (ext1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (µm)
2

Work Plane 2 (wp2)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Work Plane 2 (wp2)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

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

1 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`.

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

1 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`.

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

1 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`.

Work Plane 2 (wp2)>Fillet 1 (fil1)

1 In the **Work Plane** toolbar, click **Fillet**.
2 Click the **Zoom Extents** button in the **Graphics** toolbar.
3 On each of the rectangles **r1**, **r2**, and **r3**, in turn, select all four corner points.
4 In the **Settings** window for **Fillet**, locate the **Radius** section.
5 In the **Radius** text field, type `d/3`.

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

1 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`.

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

1 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`.

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

1 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`.

Extrude 2 (ext2)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (µm)
2

4 Select the **Reverse direction** check box.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

5 Click **Build Selected**.

Explicit Selection 1 (sel1)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.
2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 On the object **uni1**, select Boundary 10 only.
5 Locate the **Resulting Selection** section. Clear the **Keep selection** check box.
6 Find the **Cumulative selection** subsection. Click **New**.
7 In the **New Cumulative Selection** dialog box, type `Ground` in the **Name** text field.
8 Click **OK**.

Explicit Selection 2 (sel2)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.
2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 On the object **uni1**, select Boundary 29 only.
5 Locate the **Resulting Selection** section. Clear the **Keep selection** check box.
6 Find the **Cumulative selection** subsection. Click **New**.
7 In the **New Cumulative Selection** dialog box, type `Applied Voltage` in the **Name** text field.
8 Click **OK**.

Explicit Selection 3 (sel3)

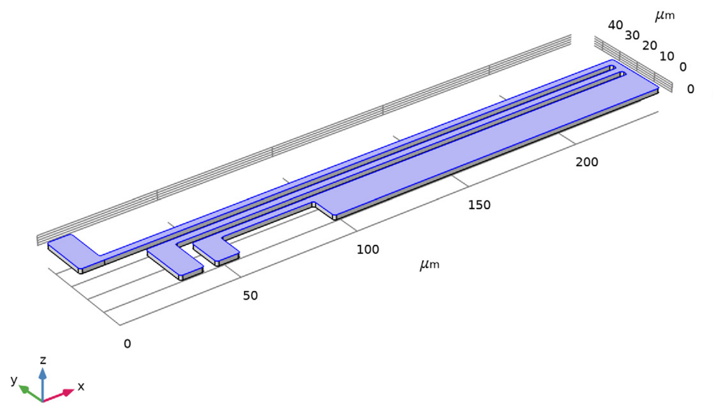
1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.
2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 On the object **uni1**, select Boundary 48 only.
5 Locate the **Resulting Selection** section. Clear the **Keep selection** check box.
6 Find the **Cumulative selection** subsection. Click **New**.
7 In the **New Cumulative Selection** dialog box, type `Third` in the **Name** text field.
8 Click **OK**.

Explicit Selection 4 (sel4)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.
2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 On the object **uni1**, select Boundary 4 only.



5 Locate the **Resulting Selection** section. Clear the **Keep selection** check box.

6 Find the **Cumulative selection** subsection. Click **New**.

7 In the **New Cumulative Selection** dialog box, type `Upper Surface` in the **Name** text field.

8 Click **OK**.

Explicit Selection 5 (sel5)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 On the object **uni1**, select Boundaries 1–3, 5–9, 11–28, 30–47, and 49–92 only.

5 In the **Model Builder** window, click **Explicit Selection 5 (sel5)**.

6 Locate the **Resulting Selection** section. Clear the **Keep selection** check box.

7 Find the **Cumulative selection** subsection. Click **New**.

8 In the **New Cumulative Selection** dialog box, type `Other Surface` in the **Name** text field.

9 Click **OK**.

Explicit Selection 6 (sel6)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

3 From the **Geometric entity level** list, choose **Point**.

4 On the object **uni1**, select Point 154 only.

5 Locate the **Resulting Selection** section. Clear the **Keep selection** check box.

6 Find the **Cumulative selection** subsection. Click **New**.

7 In the **New Cumulative Selection** dialog box, type `Tip` in the **Name** text field.

8 Click **OK**.

Explicit Selection 7 (sel7)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 On the object **uni1**, select Boundaries 67, 72, and 77 only.

5 Locate the **Resulting Selection** section. Clear the **Keep selection** check box.

6 Find the **Cumulative selection** subsection. Click **New**.

7 In the **New Cumulative Selection** dialog box, type `Dimples` in the **Name** text field.

8 Click **OK**.

End If 1 (endif1)

1 In the **Model Builder** window, click **End If 1 (endif1)**.

2 In the **Settings** window for **End If**, click **Build Selected**.

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
noa	2	2	Number of arms

GEOMETRY 1

If 2 (if2)

1 In the **Geometry** toolbar, click **Programming** and choose **If + End If**.

2. In the **Settings** window for **If**, locate the **If** section.
3. In the **Condition** text field, type `(noa==2)`.

Work Plane 3 (wp3)

1. In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Duplicate**.
2. Expand the **Work Plane 3 (wp3)** node.

Plane Geometry

In the **Model Builder** window, expand the **Thermal Actuator (comp1)>Geometry 1>Work Plane 3 (wp3)>Plane Geometry** node.

Work Plane 3 (wp3)>Rectangle 6 (r6), Work Plane 3 (wp3)>Rectangle 7 (r7), Work Plane 3 (wp3)>Rectangle 8 (r8)

1. In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1>Work Plane 3 (wp3)>Plane Geometry**, Ctrl-click to select **Rectangle 6 (r6)**, **Rectangle 7 (r7)**, and **Rectangle 8 (r8)**.
2. Right-click and choose **Disable**.

Extrude 3 (ext3)

1. In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Extrude 1 (ext1)** and choose **Duplicate**.
2. In the **Settings** window for **Extrude**, locate the **General** section.
3. From the **Work plane** list, choose **Work Plane 3 (wp3)**.
4. Find the **Input objects** subsection. Select the **Activate Selection** toggle button.
5. Select the object **wp3** only.
6. Click **Build Selected**.

Work Plane 4 (wp4)

1. In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Work Plane 2 (wp2)** and choose **Duplicate**.
2. Expand the **Work Plane 4 (wp4)** node.

Work Plane 4 (wp4)>Rectangle 1 (r1)

1. In the **Model Builder** window, expand the **Thermal Actuator (comp1)>Geometry 1>Work Plane 4 (wp4)>Plane Geometry** node, then click **Rectangle 1 (r1)**.
2. Right-click **Rectangle 1 (r1)** and choose **Disable**.

Extrude 4 (ext4)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Extrude 2 (ext2)** and choose **Duplicate**.
2 In the **Settings** window for **Extrude**, locate the **General** section.
3 From the **Work plane** list, choose **Work Plane 4 (wp4)**.
4 Find the **Input objects** subsection. Select the **Activate Selection** toggle button.
5 Select the object **wp4** only.
6 Click **Build Selected**.

Union 2 (uni2)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Union 1 (uni1)** and choose **Duplicate**.
2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

Explicit Selection 8 (sel8)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Explicit Selection 1 (sel1)** and choose **Duplicate**.
2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.
3 Find the **Entities to select** subsection. Select the **Activate Selection** toggle button.
4 On the object **uni2**, select Boundary 28 only.
5 Click **Build Selected**.

Explicit Selection 9 (sel9)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Explicit Selection 2 (sel2)** and choose **Duplicate**.
2 On the object **uni2**, select Boundary 10 only.
3 In the **Settings** window for **Explicit Selection**, click **Build Selected**.

Explicit Selection 10 (sel10)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Explicit Selection 4 (sel4)** and choose **Duplicate**.
2 On the object **uni2**, select Boundary 4 only.
3 In the **Settings** window for **Explicit Selection**, click **Build Selected**.

Explicit Selection 11 (sel11)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Explicit Selection 5 (sel5)** and choose **Duplicate**.

2 On the object **uni2**, select Boundaries 1–3, 5–9, 11–27, and 29–66 only.

3 In the **Settings** window for **Explicit Selection**, click **Build Selected**.

Explicit Selection 12 (sel12)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Explicit Selection 6 (sel6)** and choose **Duplicate**.

2 On the object **uni2**, select Point 108 only.

3 In the **Settings** window for **Explicit Selection**, click **Build Selected**.

Explicit Selection 13 (sel13)

1 In the **Model Builder** window, under **Thermal Actuator (comp1)>Geometry 1** right-click **Explicit Selection 7 (sel7)** and choose **Duplicate**.

2 On the object **uni2**, select Boundaries 47, 52, and 57 only.

3 In the **Settings** window for **Explicit Selection**, click **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

DEFINITIONS

Surface Contact

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

2 In the **Settings** window for **Union**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Ground**, **Applied Voltage**, and **Third**.

6 Click **OK**.

7 Right-click **Union 1** and choose **Rename**.

8 In the **Rename Union** dialog box, type `Surface Contact` in the **New label** text field.

9 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 **Built-in>Polysilicon**.

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

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

ELECTRIC CURRENTS (EC)

Electric Potential 1

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

2 In the **Settings** window for **Electric Potential**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Applied Voltage**.

4 Locate the **Electric Potential** section. In the V_0 text field, type `DV`.

Ground 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 In the **Settings** window for **Ground**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Ground**.

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**.

2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Other Surface**.

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.

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

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

Heat Flux 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Upper Surface**.

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.

4 Locate the **Heat Flux** section. 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 **Surface Contact**.

SOLID MECHANICS (SOLID)

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

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Surface Contact**.

Roller 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Roller**.

2 In the **Settings** window for **Roller**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Dimples**.

4 Click the **Show More Options** button in the **Model Builder** toolbar.

5 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation View**.

6 Click **OK**.

Linear Elastic Material 1

1 Click the **Refresh Equations** button.

2 In the **Model Builder** window, expand the **Thermal Actuator (comp1)>Solid Mechanics (solid)>Linear Elastic Material 1** node, then click **Equation View**.

3 In the **Settings** window for **Equation View**, locate the **Variables** section.

4 In the table, enter the following settings:

Name	Expression	Unit	Description	Selection	Details
solid.eXX	`uX-alphaps*(T-T0)`	1	Strain tensor, XX component	Domain 1	+operation
solid.eYY	`vY-alphaps*(T-T0)`	1	Strain tensor, YY component	Domain 1	+operation
solid.eZZ	`wZ-alphaps*(T-T0)`	1	Strain tensor, ZZ component	Domain 1	+operation

STUDY 1

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

RESULTS

Point Evaluation 1

1 In the **Results** toolbar, click **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, locate the **Selection** section.

3 From the **Selection** list, choose **Tip**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`solid.disp`	µm	`Displacement magnitude`

5 Click **Evaluate**.

Volume Maximum 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Maximum>Volume Maximum**.

2 In the **Settings** window for **Volume Maximum**, locate the **Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`T`	K	`Temperature`

5 Click **Evaluate**.

Deformation 1

1 In the **Model Builder** window, expand the **Temperature (ht)** node.

2 Right-click **Surface** and choose **Deformation**.

3 In the **Settings** window for **Deformation**, locate the **Scale** section.

4 Select the **Scale factor** check box.

5 In the associated text field, type 5.

6 In the **Temperature (ht)** toolbar, click **Plot**.

GLOBAL DEFINITIONS

Parameters 1

Continue by computing the solution for the three arms actuator geometry case.

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
noa	3	3	Number of arms

STUDY 1

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

RESULTS

Point Evaluation 1

1 In the **Model Builder** window, under **Results>Derived Values** click **Point Evaluation 1**.

2 In the **Settings** window for **Point Evaluation**, click **Evaluate**.

Volume Maximum 1

1 In the **Model Builder** window, click **Volume Maximum 1**.

2 In the **Settings** window for **Volume Maximum**, click **Evaluate**.