Created in COMSOL Multiphysics 6.1

On/Off Control of a Thermal Actuator

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.

This is a modified version of the *Thermal Microactuator Simplified* model from the COMSOL Multiphysics Application Library. In the present model, the applied current is controlled so that the actuator deflection does not exceed a given value. The on/off controller used in this example is implemented in Simulink.

Model Definition

The electro-thermal expansion problem is solved in COMSOL Multiphysics, while the control system is implemented in Simulink as an on/off controller.

The cosimulation with COMSOL Multiphysics and Simulink is set up by exporting a COMSOL Cosimulation file from the COMSOL model and then adding this file to the COMSOL Cosimulation block in the Simulink simulation diagram. The input of the block consists of the applied voltage. The block has two outputs: the tip displacement, which is used as control variable, and the maximum temperature in the actuator.

Figure 1 shows the full control simulation diagram in Simulink.

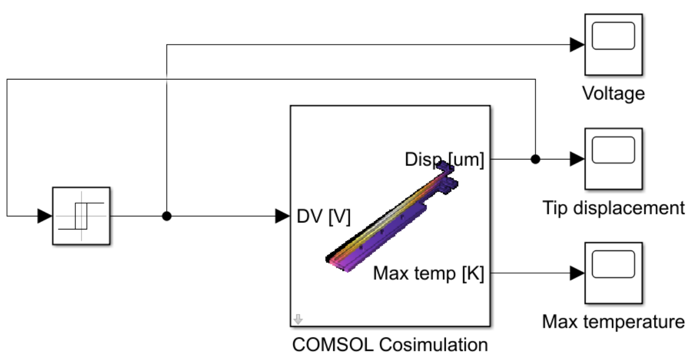

Figure 1: Simulink diagram of the thermal actuator controller.

Results and Discussion

Figure 2 below shows the actuator tip displacement at the communication time between COMSOL and Simulink. The tip displacement is always controlled between 0.71 µm and 0.81 µm. To reduce the variation in the displacement, you need to reduce the communication time and the settings in the relay block.

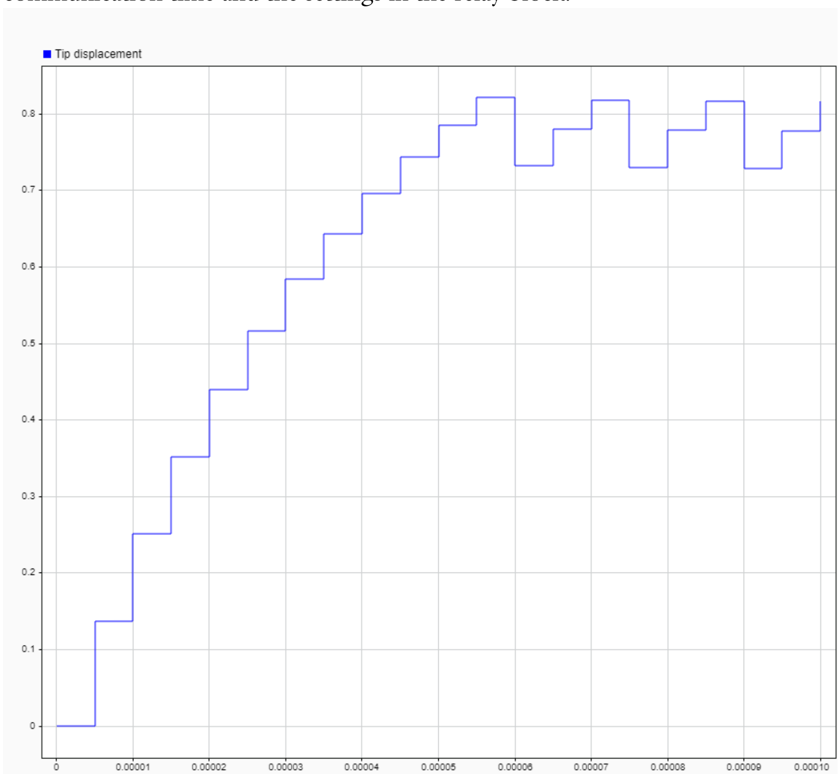

Figure 2: Actuator tip displacement at communication time.

Figure 3 below shows the actuator tip displacement that is stored in the COMSOL model once the simulation is run.

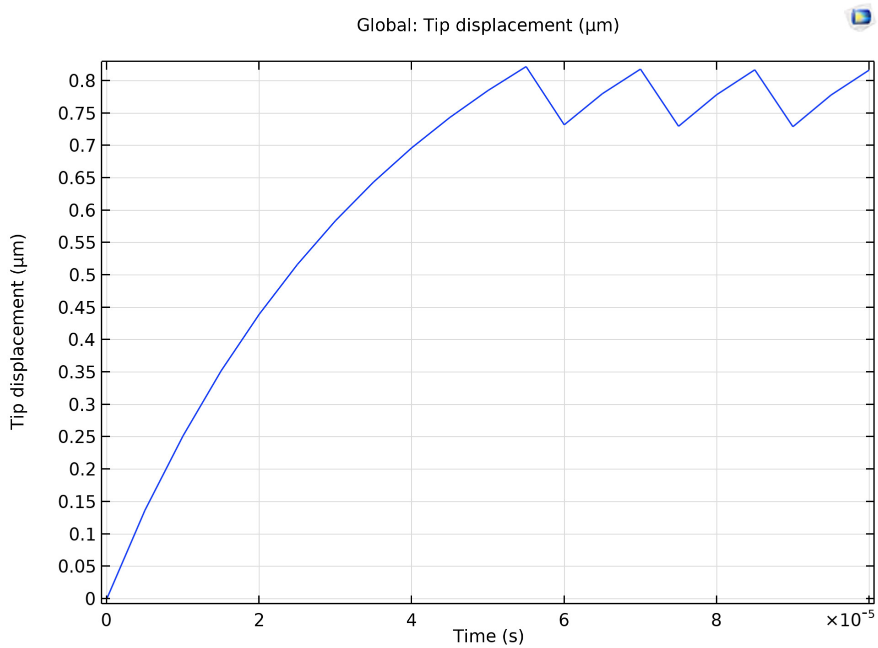

Figure 3: Actuator tip displacement as displayed in COMSOL Desktop.

Figure 4 shows the applied voltage as an input to the COMSOL model. The plot clearly shows the on/off activation based on the tip displacement of the actuator.

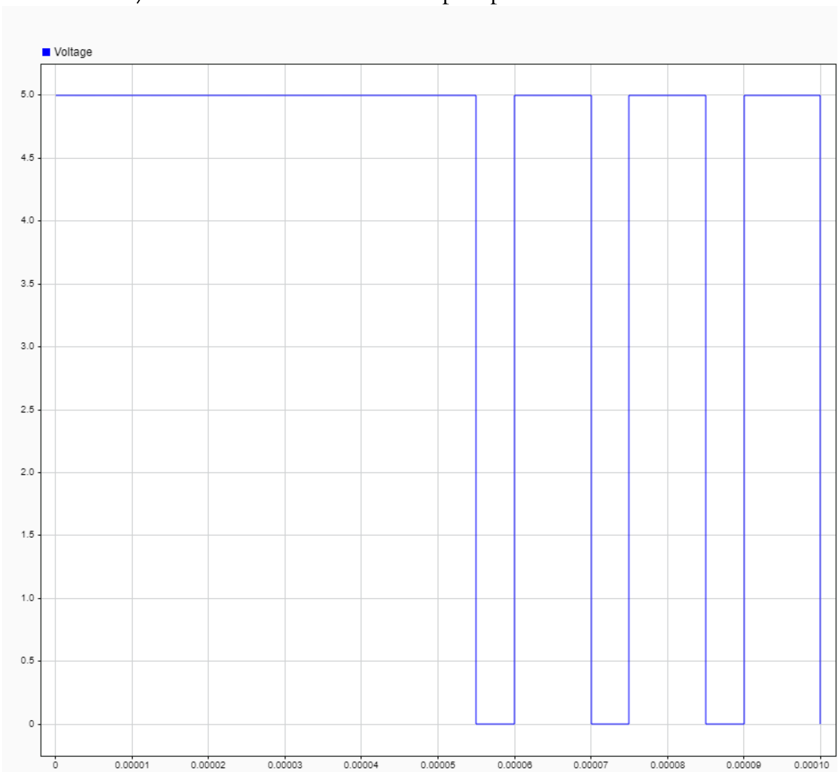

Figure 4: Applied voltage vs. time.

The maximum temperature in the actuator is shown in Figure 5. The activation intervals are too short for the cooling as the temperature keeps increasing.

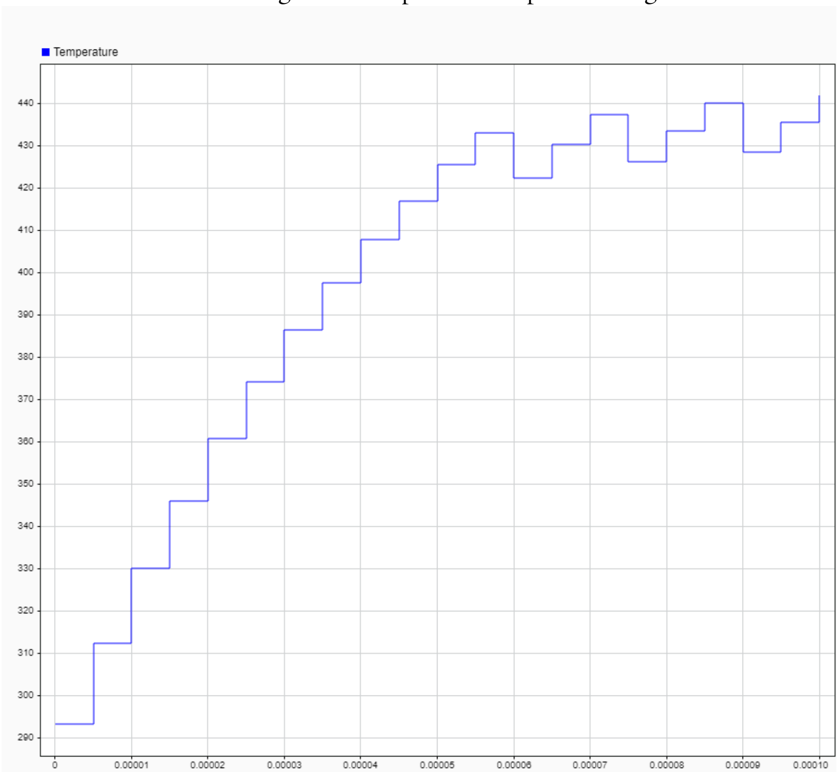

Figure 5: Actuator maximum temperature vs. time.

Setting Up the Cosimulation

Follow the workflow below to set up the cosimulation with COMSOL Multiphysics and Simulink:

1. Set up the COMSOL model and make sure that the study runs. Only studies with a single Stationary or Time Dependent study step are supported for cosimulation.
2. Save the COMSOL model. This step is important because the name of the model is needed to load the cosimulation file in Simulink.
3. Add the Cosimulation for Simulink feature node to the COMSOL model. Use this to define the inputs, outputs, and study for the cosimulation.

4 From the Cosimulation for Simulink feature node, export the file for cosimulation. Any location will work, but it is good practice to export this file to the location where the MPH-file has been saved.

5 Create or load the simulation diagram in Simulink, and add the COMSOL Cosimulation block.

6 Double-click the COMSOL Cosimulation block, and enter the name of the cosimulation file exported from COMSOL Multiphysics.

Application Library path: `LiveLink_for_Simulink/Tutorials/thermal_actuator_llsimulink`

Modeling Instructions — COMSOL Desktop

In this example you will start from an existing model which is an example in the COMSOL Multiphysics.

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **COMSOL Multiphysics>Multiphysics>thermal_actuator_simplified** in the tree.

3 Click **Open**.

DEFINITIONS

Domain Point Probe 1

1 In the **Model Builder** window, expand the **Thermal Actuator (comp1)** node.

2 Right-click **Thermal Actuator (comp1)>Definitions** and choose **Probes>Domain Point Probe**.

3 In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.

4 In row **Coordinates**, set **x** to `L`.

5 Select the **Snap to closest boundary** check box.

Point Probe Expression 1 (ppb1)

1 In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.

2 In the **Settings** window for **Point Probe Expression**, locate the **Expression** section.

3 In the **Expression** text field, type `-v`.

Domain Probe 1 (dom1)

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.

2 In the **Settings** window for **Domain Probe**, type `maxTemp` in the **Variable name** text field.

3 Locate the **Probe Type** section. From the **Type** list, choose **Maximum**.

4 Locate the **Expression** section. In the **Expression** text field, type `T`.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

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

Damping 1

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.

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

3 In the α_{dM} text field, type `1e-6`.

4 In the β_{dK} text field, type `1e-6`.

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 **Coarser**.

4 Click **Build All**.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Time Dependent**.

4 Right-click and choose **Add Study**.

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

STUDY 2

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Output times** text field, type `range(0,1e-6,1e-4)`.

3 In the **Model Builder** window, click **Study 2**.

4 In the **Settings** window for **Study**, locate the **Study Settings** section.

5 Clear the **Generate default plots** check box.

Solution 2 (sol2)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations> Solution 2 (sol2)>Dependent Variables 1** node, then click **Temperature (comp1.T)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

6 In the **Scale** text field, type `300`.

7 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)> Dependent Variables 1** click **Electric potential (comp1.V)**.

8 In the **Settings** window for **Field**, locate the **Scaling** section.

9 From the **Method** list, choose **Manual**.

10 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)** right-click **Time-Dependent Solver 1** and choose **Fully Coupled**.

11 In the **Study** toolbar, click **Compute**.

RESULTS

1D Plot Group 7

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Global 1

1 Right-click **1D Plot Group 7** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
comp1.ppb1	µm	Tip displacement

4 Click to expand the **Legends** section. Clear the **Show legends** check box.

Temperature (ht)

1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

SAVE THE COMSOL MODEL

1 From the **File** menu, choose **Save As**.

2 Browse to a suitable folder, enter the filename `thermal_actuator_llsimulink.mph`, and then click **Save**.

Exporting the Cosimulation File

In the following configure the cosimulation, and export the file for cosimulation that will be loaded into Simulink.

GLOBAL DEFINITIONS

Cosimulation for Simulink 1

1 In the **Study** toolbar, click **Cosimulation for Simulink**.

2 In the **Settings** window for **Cosimulation for Simulink**, locate the **Filename** section.

3 In the **Filename** text field, type `thermal_actuator_llsimulink`.

4 Locate the **Inputs** section. Click **Add**.

5 In the table, enter the following settings:

Parameter name	Initial value	Unit
DV (Applied voltage)	5[V]	V

6 Locate the **Outputs** section. In the table, enter the following settings:

Expression	Unit	Name
comp1.ppb1	um	Disp
comp1.maxTemp	K	Max temp

7 Locate the **Study** section. From the **Study** list, choose **Study 2**.

8 Locate the **Image** section. Click **Set from Graphics Window**This sets the current temperature plot (if a solution is available) as the thumbnail used for the COMSOL Cosimulation block inside Simulink.

9 Click **Export**.

Modeling Instructions — Simulink

Once you have created the COMSOL model and saved the cosimulation file you can start Simulink to continue with the setup there.

1 Start COMSOL with Simulink.

2 In MATLAB enter the command `mphapplicationlibraries` to start the GUI for viewing models from the LiveLink for Simulink application library.

3 Browse to the folder LiveLink_for_Simulink/Tutorials, and select thermal_actuator_llsimulink.slx.

4 Click Open to get the simulation diagram as in Figure 1.

The included COMSOL Cosimulation block is already configured with a cosimulation file based on the model from the COMSOL Application Library and ready to run. If you want to run the simulation directly, go to Step 7 below. Else, if you want to use the model file and cosimulation file you have created by following the steps in the section Modeling Instructions — COMSOL Desktop, you can continue with Step 5 below.

5 Double-click the COMSOL Cosimulation block.

6 In the COMSOL Cosimulation window settings, in the Filename edit field enter the name of the file for cosimulation for Simulink as created in the section Exporting File for Cosimulation for Simulink.

Note: In case the folder path of the file for cosimulation for Simulink is not set in MATLAB enter the full filename.

For this simulation the stop time is set to 10^{-4} s and the communication step size is set to 5 µs.

7 To run the simulation, click Run.

POSTPROCESSING THE SOLUTION IN THE COMSOL DESKTOP

To reproduce Figure 3 follow the steps below:

1 Once you have run the simulation in Simulink, go to the MATLAB prompt and enter:

```
mphlaunch
```

This will start a COMSOL Desktop with the model used to run the cosimulation.

2 In the **Model Builder**, under the **Results** node, select **1D Plot Group 7**.

Note: Close the COMSOL Desktop before running a new simulation in Simulink.
