



INTRODUCTION TO  
LiveLink™ *for* Simulink®

# Introduction to LiveLink™ for Simulink®

© 2009–2025 COMSOL

Protected by patents listed on [www.comsol.com/patents](http://www.comsol.com/patents), or see Help>About COMSOL Multiphysics on the File menu in the COMSOL Desktop for less detailed lists of U.S. Patents that may apply. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement ([www.comsol.com/comsol-license-agreement](http://www.comsol.com/comsol-license-agreement)) and may be used or copied only under the terms of the license agreement.

COMSOL, the COMSOL logo, COMSOL Multiphysics, COMSOL Desktop, COMSOL Compiler, COMSOL Server, and LiveLink are either registered trademarks or trademarks of COMSOL AB. MATLAB and Simulink are registered trademarks of The MathWorks, Inc.. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those or the above non-COMSOL trademark owners. For a list of such trademark owners, see [www.comsol.com/trademarks](http://www.comsol.com/trademarks).

Version: COMSOL 6.4

## Contact Information

Visit the Contact COMSOL page at [www.comsol.com/contact](http://www.comsol.com/contact) to submit general inquiries or search for an address and phone number. You can also visit the Worldwide Sales Offices page at [www.comsol.com/contact/offices](http://www.comsol.com/contact/offices) for address and contact information.

If you need to contact Support, an online request form is located on the COMSOL Access page at [www.comsol.com/support/case](http://www.comsol.com/support/case). Useful links:

- Support Center: [www.comsol.com/support](http://www.comsol.com/support)
- Product Download: [www.comsol.com/product-download](http://www.comsol.com/product-download)
- Product Updates: [www.comsol.com/product-update](http://www.comsol.com/product-update)
- COMSOL Blog: [www.comsol.com/blogs](http://www.comsol.com/blogs)
- Discussion Forum: [www.comsol.com/forum](http://www.comsol.com/forum)
- Events: [www.comsol.com/events](http://www.comsol.com/events)
- COMSOL Video Gallery: [www.comsol.com/videos](http://www.comsol.com/videos)
- Support Knowledge Base: [www.comsol.com/support/knowledgebase](http://www.comsol.com/support/knowledgebase)
- Learning Center: [www.comsol.com/support/learning-center](http://www.comsol.com/support/learning-center)

Part number: CM020010

# Contents

---

Introduction . . . . .	5
Starting COMSOL Multiphysics® with Simulink®. . . . .	6
A Thorough Example: On/Off Control of a Thermal Actuator. . . . .	7



# Introduction

---

LiveLink™ *for* Simulink® connects COMSOL Multiphysics® to the Simulink® system simulation environment. Using this functionality, you can do the following:

- Perform cosimulation of COMSOL models together with Simulink diagrams.
- Export state-space models based on COMSOL models. These state-space models are linearized models based on the total number of degrees of freedom (DOFs) for the COMSOL model.
- Run COMSOL models as state-space systems in Simulink diagrams. These state-space models are linearized models using either the total number of degrees of freedom or using modal order reduction.
- Use regular COMSOL and Simulink features to set up additional types of models.

The examples in this guide take you through the process of setting up a COMSOL model and explain how to use COMSOL Multiphysics within the Simulink simulation environment.

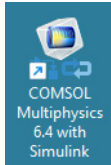
# Starting COMSOL Multiphysics® with Simulink®

---

## Starting on Windows®

---

To start COMSOL Multiphysics with Simulink, double-click the COMSOL Multiphysics 6.4 with Simulink icon available on the desktop.



This opens the MATLAB desktop along with the COMSOL Multiphysics Server, which is represented by the command window appearing in the background.

## Starting on macOS

---

Navigate to Applications > COMSOL 6.4 > COMSOL Multiphysics 6.4 with Simulink.

## Starting on Linux®

---

Start a terminal prompt and run the `comsol` command, which is located inside the `bin` folder in the COMSOL installation directory:

```
comsol mphserver simulink
```

# A Thorough Example: On/Off Control of a Thermal Actuator

---

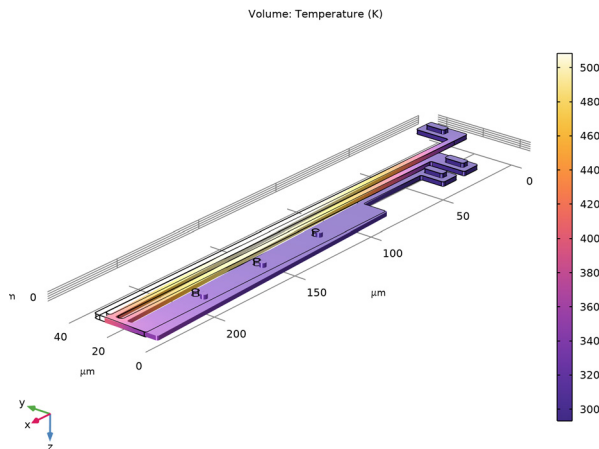
This tutorial familiarizes you with the workflow to set up a cosimulation between the COMSOL model and a Simulink diagram.

In this section, learn how to:

- Prepare the COMSOL model for a cosimulation
- Export the cosimulation file
- Include a COMSOL Cosimulation block in a Simulink diagram
- Run the cosimulation
- Work with the COMSOL model after the simulation

The example is based on a modified version of the Thermal Microactuator Simplified model from the COMSOL Multiphysics application library, which analyzes the electrothermal deformation of a microactuator.

This 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.



*Temperature distribution in a deformed shape of the microactuator.*

The original model focuses on obtaining the stationary solution for the actuator deformation due to resistive heating (Joule heating). In this tutorial, we modify





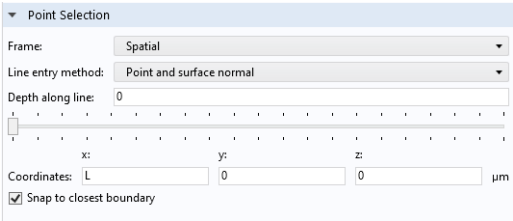
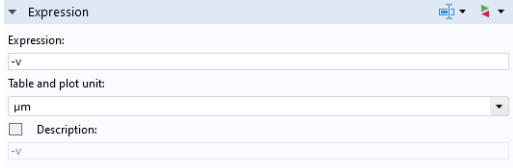
the model to include dynamical effects, and control the applied current to maintain the actuator deflection within set limits.



**Note:** The step-by-step instructions below are designed to be carried out in sequence. Skipping any of the sections might result in data being unavailable for the following sections. Start with [Setting Up the COMSOL Model for Cosimulation](#) and work through the sections until reaching the last section, [Working With the Model After the Cosimulation](#).

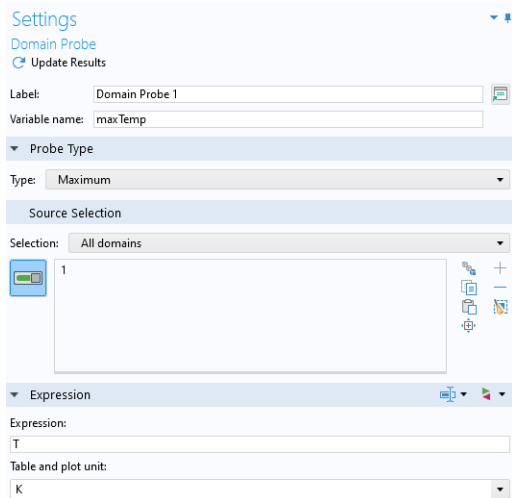
## Setting Up the COMSOL Model for Cosimulation

---


Load the model *Thermal Microactuator Simplified* from the COMSOL Multiphysics Application Library.

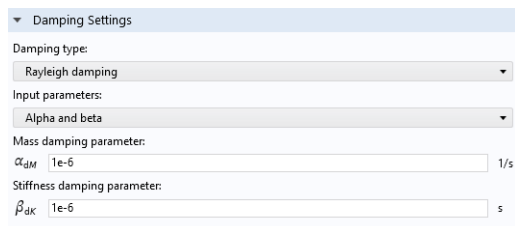
- 1 If it is not already open, start a new COMSOL Desktop. From the File menu, select Application Libraries (  ).
- 2 In the Application Libraries window, choose COMSOL Multiphysics>Multiphysics>thermal\_actuator\_simplified and click Opens (  ). Start with setting up probes in the model to evaluate the actuator tip displacement and the maximum temperature in the actuator. Further ahead, when configuring the cosimulation, you will set these as the output that is passed to Simulink during cosimulation.
- 3 In the Definitions toolbar click Probes , and choose Domain Point Probe  .
- 4 In the Settings window for Domain Point Probe, under Point Selection section, in the Coordinates row set x to L. The screenshot shows the 'Point Selection' settings window. The 'Frame' is set to 'Spatial'. The 'Line entry method' is 'Point and surface normal'. The 'Depth along line' is set to 0. The 'Coordinates' row has 'x' set to 'L', 'y' set to '0', and 'z' set to '0'. The unit is 'µm'. The 'Snap to closest boundary' checkbox is checked.
- 5 Select the Snap to closest boundary checkbox.
- 6 In the Model Builder window, expand the Domain Point Probe 1 node, then click Point Probe Expression 1 (ppbb1).
- 7 In the Settings window for Point Probe Expression, in the Expression text field, type -v. The screenshot shows the 'Expression' settings window. The 'Expression' field contains '-v'. The 'Table and plot unit' is set to 'µm'. The 'Description' field contains '-v'.

- 8 In the Definitions toolbar click Probes , and choose Domain Probe .
- 9 In the Settings window for Domain Probe, in the Variable name field, type `maxTemp`.
- 10 In the section Probe Type, select Maximum as Type.
- 11 In the Expression text field, type `T`.







Continue with adding structural damping to the Solid Mechanics interface.

- 12 In the Model Builder window, expand the Thermal Actuator (comp1) > Solid Mechanics (solid) node, then click Linear Elastic Material 1.
- 13 In the Physics toolbar, click Attributes  and choose Damping.
- 14 In the Settings window for Damping, in both the  $\alpha_{dM}$  and  $\beta_{dK}$  text fields enter  $1e-6$ .




Now continue with the setup of the time-dependent study that you will use for the cosimulation.



- 15 In the Home toolbar, click Windows  and choose Add Study .
- 16 In the Add Study window, find the Studies subsection.
- 17 In the Select Study tree, select General Studies>Time Dependent , and then click Add Study .
- 18 In the Settings window for Time Dependent, in the Output times text field, type `range(0, 1e-6, 1e-4)` as we assume the time unit in Simulink to be seconds.

**Note:** Make sure to use the same time units in COMSOL and Simulink, as there is no time unit conversion when running a cosimulation.

Continue with setting up manual scaling for the variables to avoid potential convergence problems due to the use of an on/off controller during simulation.

- 19 In the Settings window for Study 2, clear Generate default plots.
- 20 In the Model Builder right-click the Study 2 node, and select Show Default Solver .
- 21 Expand Solution 2 > Dependent Variables 1.
- 22 Select the Temperature (comp1.T) node. In the Settings window for Field, locate the Scaling section and set Method to Manual. In the Scale text field enter 300.
- 23 Select the Electric potential (comp1.V) node. In the Settings window for Field, locate the Scaling section and set Method to Manual. The default value already corresponds to the order of magnitude of the voltage so you do not need to change the value.

It is good practice to make sure that the model solves with the intended time interval, time steps, and input parameter range that will be used when performing the cosimulation.

- 24 In the Study toolbar, click Compute  .  
The model is now ready for cosimulation, you just need to save it with a new name.
- 25 From the File menu, choose Save As .
- 26 Browse to a suitable folder, enter the filename `thermal_actuator_11simulink.mph`, and then click Save.

## Configuring and Exporting the Cosimulation File

---

You can continue with configuring the cosimulation and exporting the cosimulation file that will be loaded into Simulink. The most important cosimulation settings are:

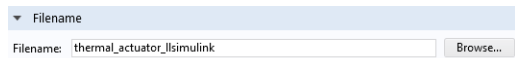
- The model name that is set when you save the model MPH file.
- The list of inputs. These are the model parameters that can be updated by Simulink during the cosimulation. This list can remain empty.
- The list of block parameters. These model parameters can be updated by Simulink when initializing the cosimulation, for example to define the initial conditions. This list can remain empty.

- The list of outputs. These are the expressions, defined in probes, that are evaluated and returned to Simulink during cosimulation. Cosimulation requires at least one output.
- The study to use for the cosimulation. You can select study nodes with a single time dependent or stationary study step.


You define the cosimulation settings in a Cosimulation for Simulink node that is added under Global Definitions in the Model Tree. You can add several Cosimulation for Simulink nodes to a model, for example to save different cosimulation settings for use with different Simulink diagrams.

1 In the Study toolbar, click Cosimulation for Simulink .

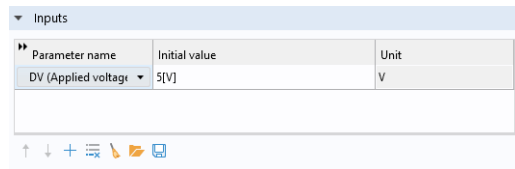
2 In the Settings window for Cosimulation for Simulink, under Filename section in the Filename text field, type `thermal_actuator_1lsimulink`.



**Note:** The name of the cosimulation file is independent of the model filename.

3 Locate the Inputs section. Click Add .

4 In the Parameter name list select DV (Applied voltage).



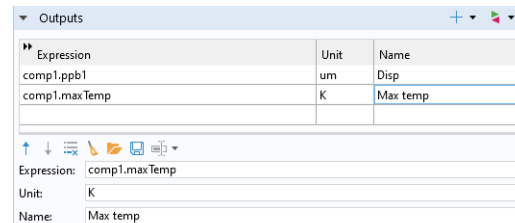
**Note:** Inputs can vary during simulation the cosimulation, hence parameters that affect the geometry or mesh are not allowed.

**Note:** Make sure that the units for the inputs are the same in the model and in Simulink as there is no unit conversion.

As there are no block parameters in this tutorial, continue with defining the outputs: the displacement at the tip of the actuator evaluated using the Domain Point Probe (`comp1.ppb1`) and the maximum temperature of the actuator evaluated using the `maxop1` operator.

5 In the first row of the Outputs table, in the Expression column enter `comp1.ppb1`, in the Unit column enter `um`, and in the Name column enter `Disp`.

6 In the second row, in the Expression column enter `comp1.maxTemp` and in the Name column enter `Max temp`.

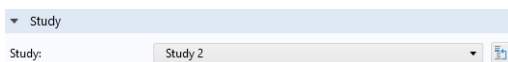


**Note:** The text you enter in the Name column will be displayed for the corresponding output in the COMSOL Cosimulation block after the cosimulation file is loaded by the block. You may leave this column empty.

For time dependent study you need to specify the case of direct feedthrough, that is, when the output expressions are function of the inputs. This is important as direct feedthrough affects the execution order of the COMSOL Cosimulation block in Simulink or may induce algebraic loops. Direct feedthrough are defined under the Dependencies section by using the radio buttons.

In this model there is no direct dependencies between the input voltage and the actuator tip displacement or the maximum temperature, so you can go to the next section.

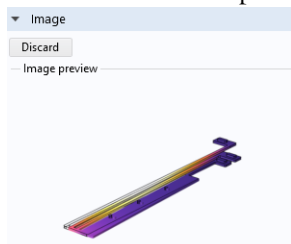
- 7 Locate the Study section. From the Study list, choose Study 2, which is the Time Dependent study you have just added to the model.




**Note:** The Study list contains only the allowed Study nodes for the cosimulation. If the list is empty, check that the model contains a study with a single stationary or time dependent study step, and that the study step does not include auxiliary sweep, load cases, or adaptation.

As the last step of setting up the cosimulation, choose the image that is displayed in the COMSOL Cosimulation block.

- 8 Under the Results node, click Temperature (ht) node to display the temperature plot in the Graphics window.
- 9 Under the Global Definitions node, click the Cosimulation for Simulink node.
- 10 In the Settings window for Cosimulation for Simulink, in the Image section, click Set from Graphics Window.



This image will be used as an icon in the COMSOL Cosimulation block. You can use any plot from the Graphics window or import your own icon. To import your own icon, browse to the desired file and then click Import.

- To conclude the cosimulation set up, click Export , available at the top of the Cosimulation for Simulink Settings window, to save the cosimulation file to the same location as the model MPH file.

After the export, the Filename text field displays the full file path for the cosimulation file.

- From the File menu choose Save to save the model, then close COMSOL Multiphysics.

In the following you will load the cosimulation file in Simulink and run the simulation.

## Adding the COMSOL Cosimulation Block

---

- Start COMSOL with Simulink, as described in [Starting COMSOL Multiphysics® with Simulink®](#).

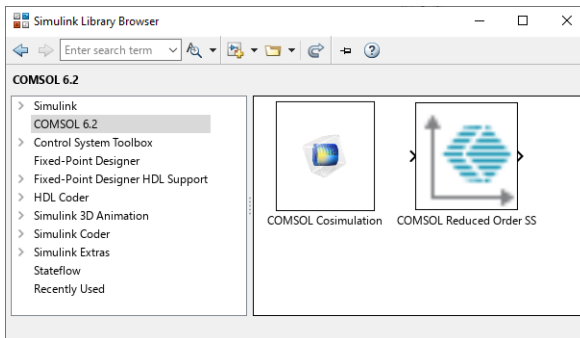
This starts a COMSOL Multiphysics server and the MATLAB prompt, where the LiveLink for Simulink welcome message is displayed.

Before you start Simulink make sure that the folder where you have just saved the model MPH file is set in MATLAB.

- In the MATLAB Home toolbar, press the Simulink button to start the Simulink environment.
- In the Simulink Start Page window, locate the Blank Model section and click Create Model.

Proceed by adding a COMSOL Cosimulation block to the diagram.

- In the Simulink Simulation toolbar, click Library Browser.
- In the Simulink Library Browser window, select COMSOL 6.4 from the list to access the COMSOL block library.

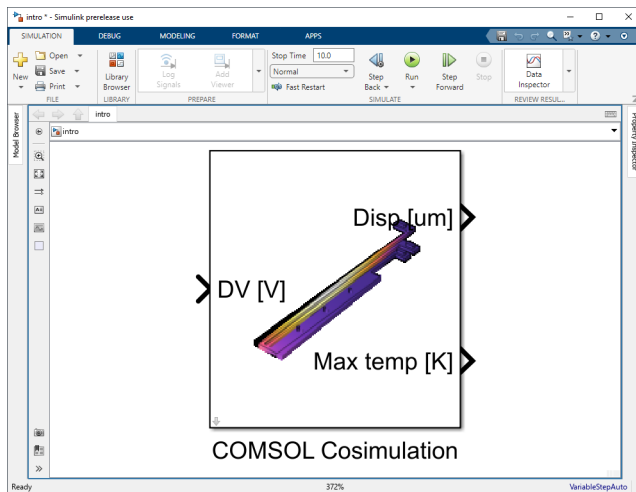


- 6 Drag and drop the COMSOL Cosimulation block to the Simulink diagram.
- 7 Double-click the COMSOL Cosimulation block to access the block settings.
- 8 In the Block Parameters: COMSOL Simulation window, in the Filename field, enter the full filepath of the cosimulation file as created in the section [Configuring and Exporting the Cosimulation File](#).

**Note:** If the cosimulation file is stored in folder, which path is set in MATLAB, you only need to enter the name of the file in the Filename field.

- 9 In the Communication step size field enter:  $5e-6$ . This means that COMSOL and Simulink will communicate every 5  $\mu$ s.

10 Click OK.



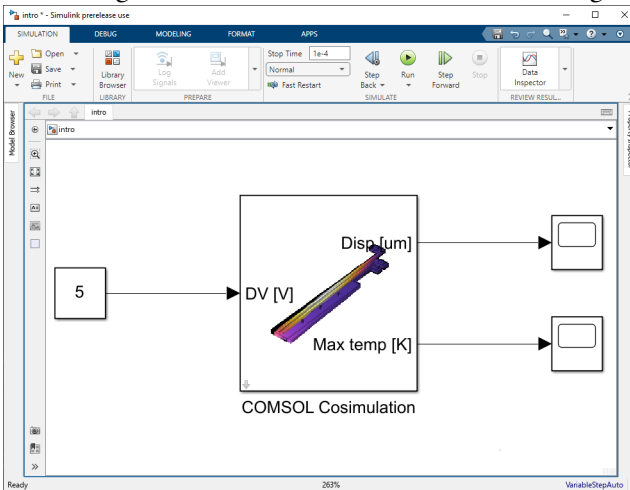
The block is now updated according to the cosimulation file settings. Here we recognize the single input parameter DV, and the two outputs named disp and max temp.

Between bracket you can see the units for the block input and outputs. These are for information purpose to let you the input of the expected input value and the returning output values.

**Note:** There is no support unit transformation inside the COMSOL Cosimulation block.

- 11 In the Simulink diagram, add a Constant block set to 5 and connect it to the COMSOL Cosimulation block input. This sets the input voltage to 5 V.
- 12 Now add scopes for the two outputs. This way you can monitor the tip displacement and the maximum temperature.

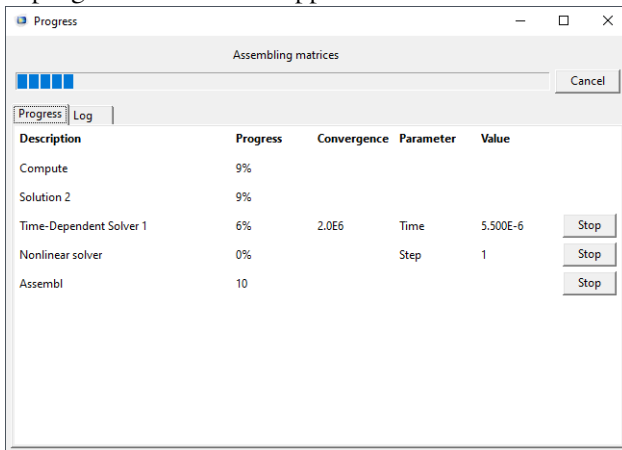
- In the Stop Time text field, enter  $1e-4$ . This sets the simulation time to 10 ms. The diagram should now look like one in the image below:



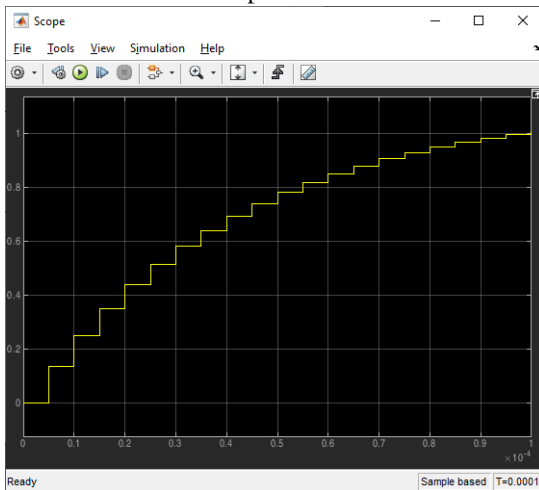
## Running the Cosimulation

- In the Simulation toolbar, click Run to start the simulation. You can monitor the progress in Simulink by watching the progress bar and the updates of the scope. On Windows<sup>®</sup> and Linux<sup>®</sup> operating systems, you can

monitor the progress in COMSOL for each communication step in the solver progress window that appears.



The scope in the following image shows the displacement in micrometers. Since the communication time step is set to  $5 \times 10^{-6}$  seconds the curve appears to have steps. It is an important part of the modeling procedure to choose a suitable communication step size to obtain a reasonable accuracy.



We will now replace the constant voltage with an on/off controller to maintain the tip displacement between  $0.71 \mu\text{m}$  and  $0.81 \mu\text{m}$ .

- 2 Delete the Constant block from the Simulink diagram.

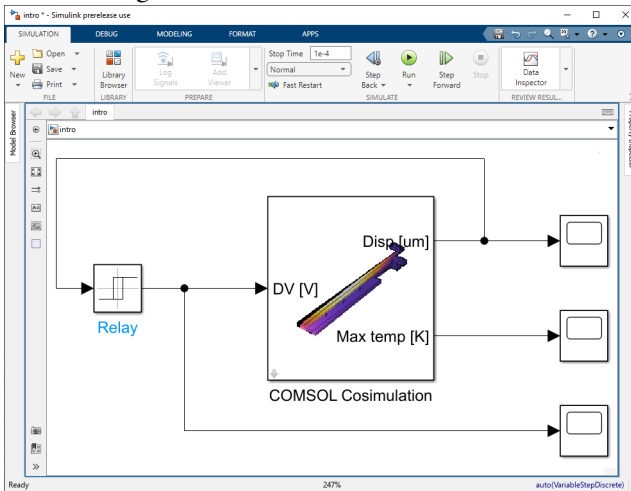
3 Add a Relay block, and set the following parameters:

PARAMETERS	VALUES
Switch on point	0.81
Switch off point	0.79
Output when on	0
Output when off	5

4 Now link the COMSOL Cosimulation block output port 1(disp) to the Relay block input port and the Relay block output port to the COMSOL Cosimulation block input port.

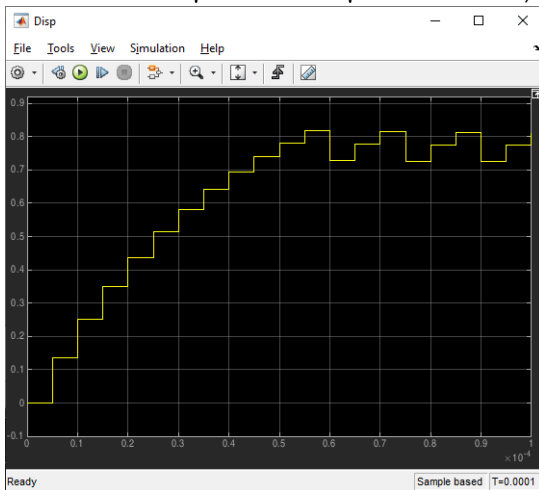
5 Finally add a scope to display the Relay block output port.

You can also change the name of the scope so that the Simulink diagram looks like the figure below.

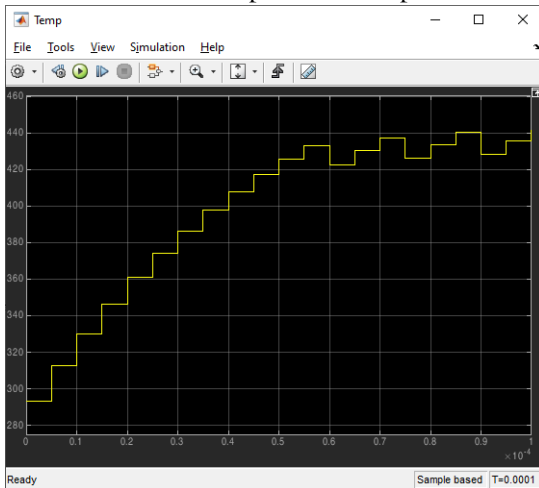


6 In the Simulation toolbar, click Run to start the simulation.

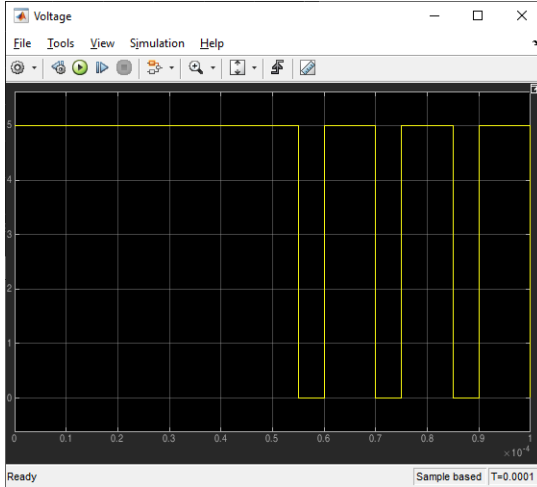
The scope for the tip displacement now shows that the displacement is kept between  $0.81\ \mu\text{m}$  and  $0.79\ \mu\text{m}$  after the on/off controller has been activated.



The maximum temperature is kept between 420 K and 440 K.



The voltage scope shows the activation of the actuator.



## Working With the Model After the Cosimulation

---

Once you have performed the solution in Simulink, you may want to postprocess the solution of the full COMSOL model and not only on the variables you have access to from the Simulink diagram.

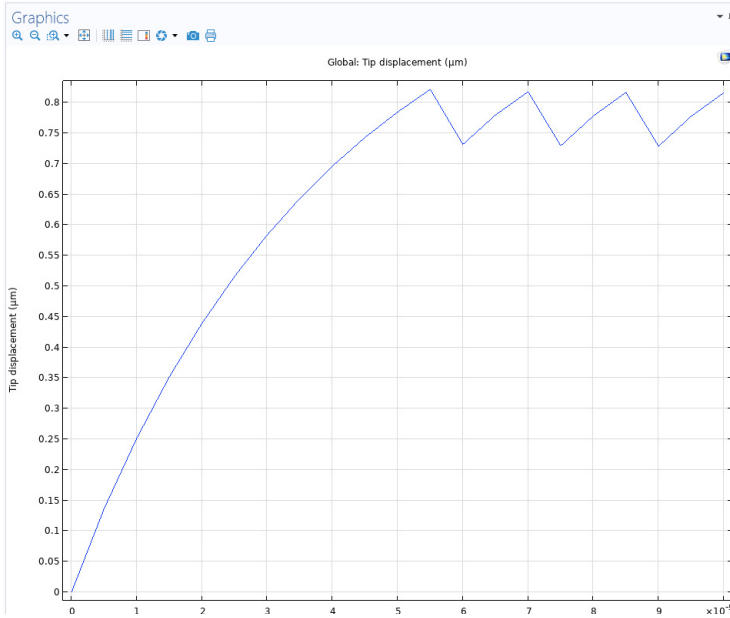
After the Simulink simulation is completed the model used for the cosimulation is still loaded on the COMSOL Multiphysics Server. You can launch the COMSOL Desktop from MATLAB to view and postprocess the model.

- 1 Switch to MATLAB.
- 2 At the MATLAB prompt enter `mphlaunch` to launch the COMSOL Desktop that automatically loads the model object from the COMSOL Multiphysics server.

**Note:** By default the model is loaded every time you run a simulation, so the modification you make to the model in the COMSOL Desktop will not affect the Simulink simulation unless you have saved the model MPH file.

Continue in the COMSOL Desktop and plot the tip displacement.

### 3 Under the Results node select the 1D Plot group 7



**Note:** If you plan to run a new Simulink simulation with the current model, it is recommended that you close the COMSOL Desktop first. Closing COMSOL Desktop leads to a smoother and often faster cosimulation.