

# INTRODUCTION TO LiveLink™ *for* Simulink®

# Introduction to LiveLink™ for Simulink®

© 2009–2020 COMSOL

Protected by patents listed on [www.comsol.com/patents](http://www.comsol.com/patents), and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,457,932;; 9,098,106; 9,146,652; 9,323,503; 9,372,673; 9,454,625, 10,019,544, 10,650,177; and 10,776,541. 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 5.6

## Contact Information

Visit the Contact COMSOL page at [www.comsol.com/contact](http://www.comsol.com/contact) to submit general inquiries, contact Technical Support, 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 at the COMSOL Access page at [www.comsol.com/support/case](http://www.comsol.com/support/case). Other useful links include:

- 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/support/updates](http://www.comsol.com/support/updates)
- COMSOL Blog: [www.comsol.com/blogs](http://www.comsol.com/blogs)
- Discussion Forum: [www.comsol.com/community](http://www.comsol.com/community)
- Events: [www.comsol.com/events](http://www.comsol.com/events)
- COMSOL Video Gallery: [www.comsol.com/video](http://www.comsol.com/video)
- Support Knowledge Base: [www.comsol.com/support/knowledgebase](http://www.comsol.com/support/knowledgebase)

Part number: CM020010

# Contents

---

Introduction .....	4
Starting COMSOL Multiphysics® with Simulink®. ....	5
A Thorough Example: On/Off Control of a Thermal Actuator. ....	6



# Introduction

---

LiveLink™ *for* Simulink® connects COMSOL Multiphysics® to the Simulink 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 system in Simulink diagrams. These state-space models are linearized models using either the total number of degrees of freedom or using modal order reduction.
- In addition it is possible to use regular COMSOL and Simulink features to set up additional types of model e.g. based on lookup tables in Simulink.

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.

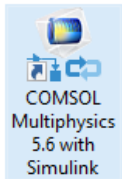
# Starting COMSOL Multiphysics® with Simulink®

---

## Starting on Windows®

---

To start COMSOL Multiphysics with Simulink, double-click the COMSOL Multiphysics 5.6 with Simulink icon available on the desktop.



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

## Starting on Mac OS X

---

Navigate to Applications>COMSOL 5.6>COMSOL Multiphysics 5.6 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

---

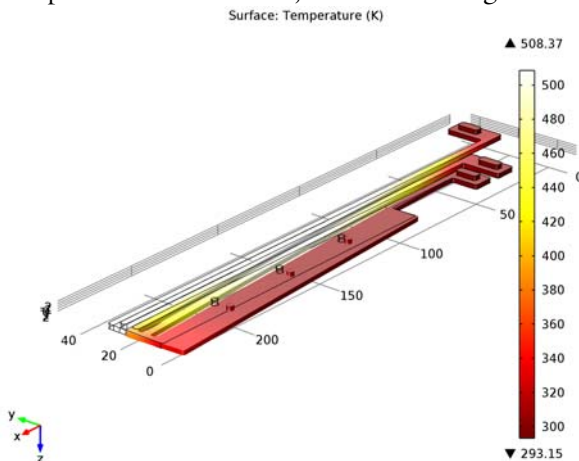
This tutorial familiarizes you with the workflow to set up a cosimulation between COMSOL model and a Simulink diagram.

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 model, Thermal Microactuator Simplified, from the COMSOL Multiphysics application library, which analyzes the electro-thermal 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 (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 a 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 Model After the Cosimulation](#).

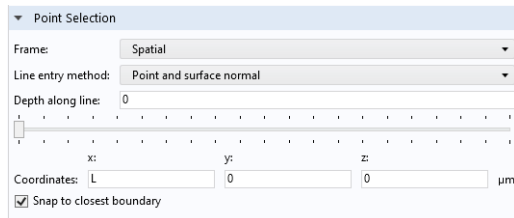
## Setting Up the COMSOL Model For Cosimulation

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 toolbar, select Application Libraries (  ).
- 2 In the Application Libraries window, choose COMSOL Multiphysics>Multiphysics>thermal\_actuator\_simplified and click Open Application.

Start with setting up a probe to evaluate the actuator tip displacement, then create a maximum operator to obtain the maximum temperature. Further ahead, when configuring the cosimulation, you will set these as the output that is passed to Simulink during cosimulation.

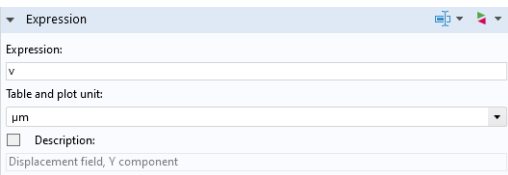
- 3 In the Definitions toolbar click Probes , and choose Domain Point Probe .



- 4 In the Settings window for Domain Point Probe, under Point Selection section, in the Coordinates row set x to L.

- 5 Select the Snap to closest boundary check box.

- 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.

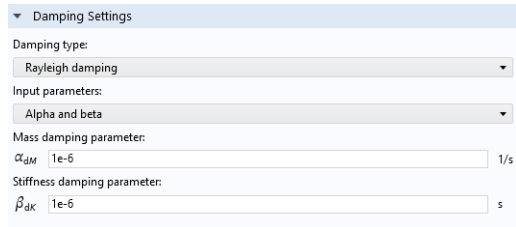
- 8 In the Definitions toolbar click Nonlocal Couplings , and select Maximum **MAX**.




- 9 In the Settings window for Maximum, from the Selection list, choose All domains.

Continue with adding structural damping to the Solid Mechanics physics interface.



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





- 11 In the Physics toolbar, click Attributes  and choose Damping.

- 12 In the Settings window for Damping, in both the  $\alpha_{dM}$  and  $\beta_{dM}$  text fields enter  $1e-6$ .

Now continue with the set up of the time dependent study that you will use for the cosimulation.

- 13 In the Home toolbar, click Windows  and choose Add Study .


- 14 In the Add Study window, find the Studies subsection.

- 15 In the Select Study tree, select General Studies>Time Dependent , and click Add Study .

- 16 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.

- 17 In the Model Builder right-click the Study 2 node, and select Show Default Solver .

- 18 Expand Solution 2 > Dependent Variables 1.

- 19 Select the Temperature (comp1.T) node. In the Settings window for Field, locate the Scaling section and set Method to Manual. In the Scale edit field enter 100.


- 20 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 don't 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.

- 21 In the Study toolbar, click Compute .

The model is now ready for cosimulation, you just need to save it with a new name.

- 22 From the File menu, choose Save As .

- 23 Browse to a suitable folder, enter the filename `thermal_actuator_11simulink.mph`, and then click Save.

## Configuring and Exporting the Cosimulation File

---

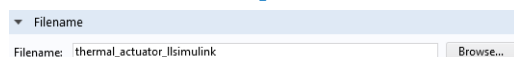
You can continue with configuring the cosimulation and exporting the cosimulation file that will be loaded into Simulink. The most important cosimulation settings are:

- 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 variables that are evaluated and returned to Simulink during cosimulation. You may define any expressions of real scalar global variables. Cosimulation requires at least one output.
- The study to use for the cosimulation. You may 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 may 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_11simulink`.

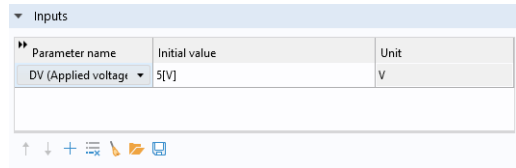


**Note:** The name of the cosimulation file is independent of the model file name.

- 3 Locate the Inputs section.

Click Add .

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



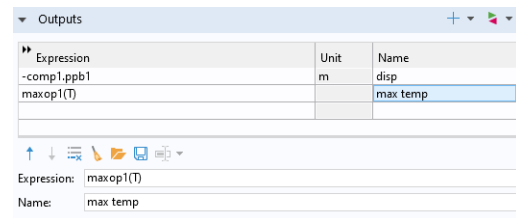
Parameter name	Initial value	Unit
DV (Applied voltage)	5[V]	V

**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` and in the Name column enter `disp`.

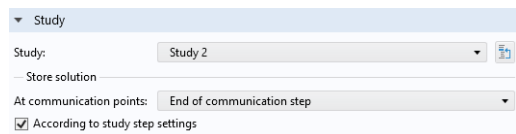


Expression	Unit	Name
-comp1.ppb1	m	disp
maxop1(T)		max temp

- 6 In the second row, in the Expression column enter `maxop1(T)` 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.

- 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.

The Study section also contains the settings for saving the solution data in the model during a cosimulation. By default the solution is stored at the end of each

communication step as well as any output times specified in the time dependent solver settings.

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 Global Definitions node, click Cosimulation for Simulink node.
- 10 In the Settings window for Cosimulation for Simulink in the Image section, click Set from Graphics Window.



- 11 To conclude the cosimulation set up, click Export  to save the cosimulation file to the same location as the model MPH-file.

After the export the filename edit field displays the full file path for the cosimulation file.

- 12 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

---

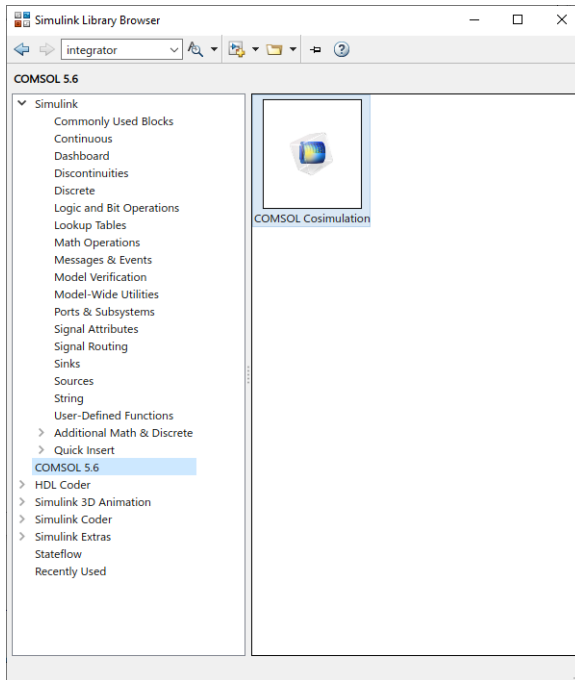
- 1 Start COMSOL with Simulink, as described in [Starting COMSOL Multiphysics® with Simulink®](#).

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

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

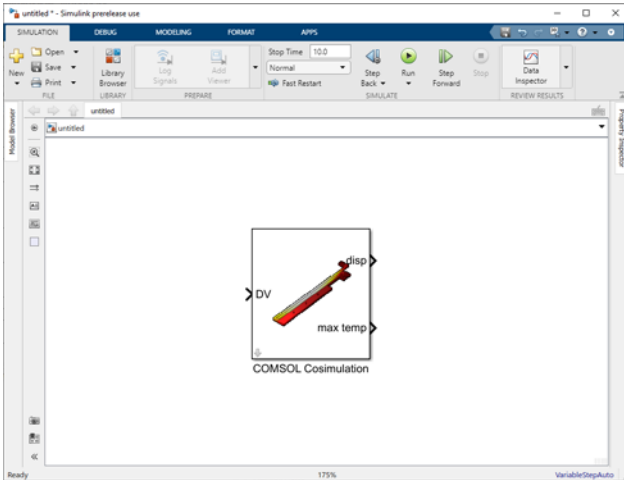
Proceed by adding a COMSOL Cosimulation block to the diagram.

- 4 In the Simulink Simulation toolbar, click Library Browser.
- 5 In the Simulink Library Browser window, from the list on left-hand side and choose COMSOL 5.6 to access the COMSOL block library.



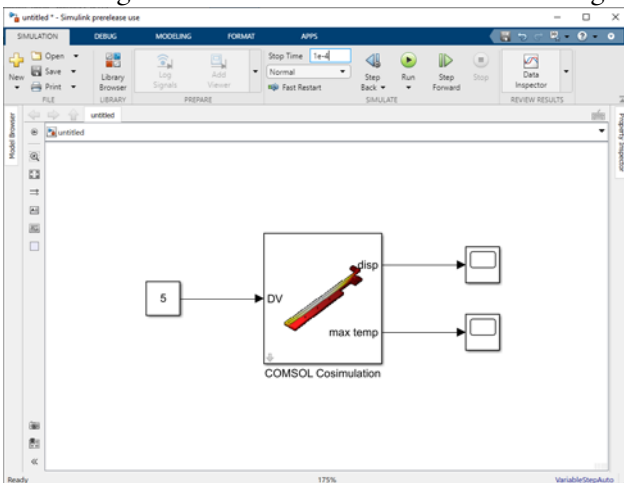
- 6 Drag and drop the COMSOL Cosimulation block to the Simulink diagram.
- 7 Double-click the COMSOL Cosimulation block to access the block settings.
- 8 In the Block Parameters: COMSOL Simulation window in the Filename edit field enter the full filepath of the cosimulation file as created in the section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

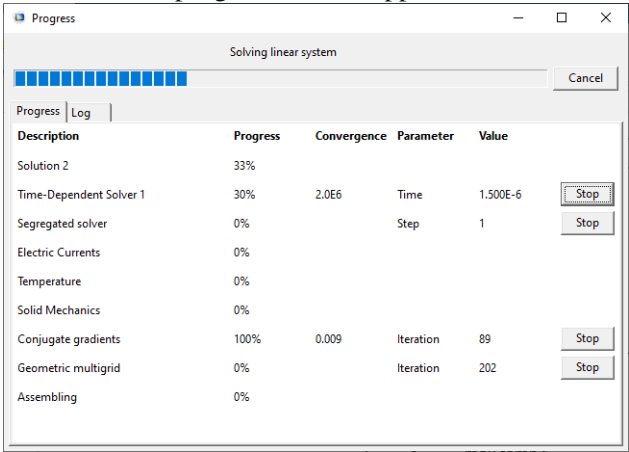
- 11** In the Simulink diagram add a Constant block set to 5 and connect it to the COMSOL Cosimulation block input. This sets the input voltage to 5 V.
- 12** Now add scopes for the two outputs. This way you can monitor the tip displacement and the maximum temperature.
- 13** In the Stop Time edit field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

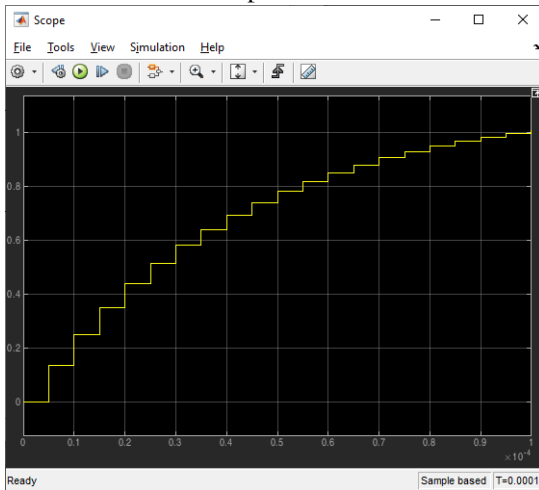
| In the Simulation toolbar, click Run to start the simulation.

After a short while the model is loaded and initialized, and the simulation will commence. You can monitor the progress in Simulink by monitoring the progress bar and watching the updates of the scope. On Windows® and Linux® OS, you can monitor the progress in COMSOL for each communication step in the solver progress bar that appears.



The scope in the following image shows the displacement in micrometers. Since the communication time step is set to 5e-6 seconds the curve appears to have

steps. It is an important part of the modeling procedure to choose a communication step size to obtain a reasonable accuracy.



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

- 2 Delete the Constant block from the Simulink diagram.
- 3 Add a Relay block, and set the following parameters:

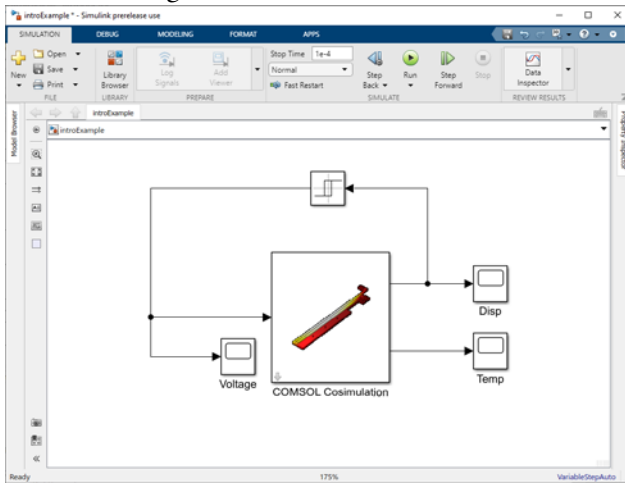
PARAMETERS	VALUES
Switch on point	0.81
Switch off point	0.79
Output when on	0
Output when off	5

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



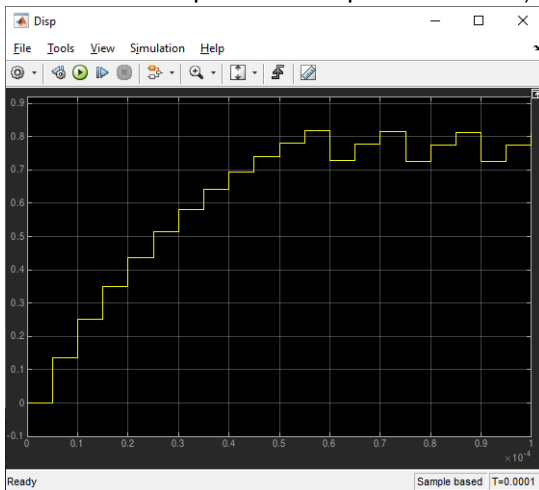
5 Finally add a scope to display the Relay block output port.

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

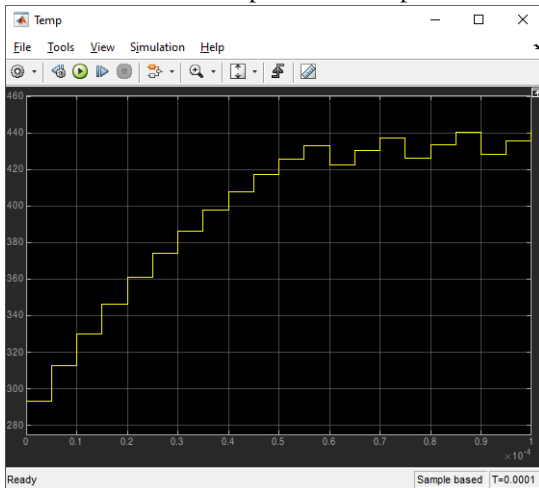


6 In the Simulation toolbar, click Run to start the simulation.

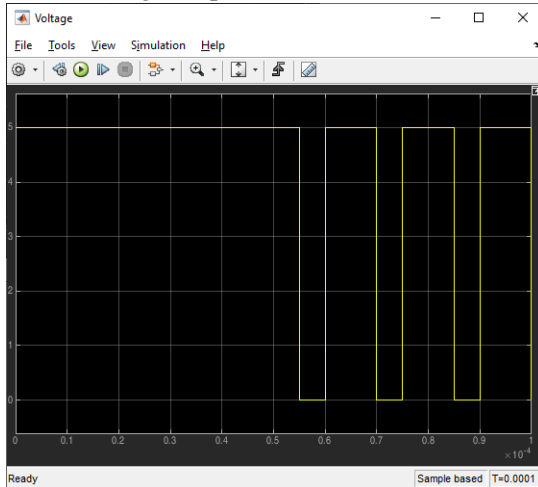
The scope for the tip displacement now shows that the displacement is kept between  $0.81\text{ }\mu\text{m}$  and  $0.79\text{ }\mu\text{m}$  after the on/off controller has been activated.



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



The voltage scope shows the activation of the actuator.



## Working With 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.

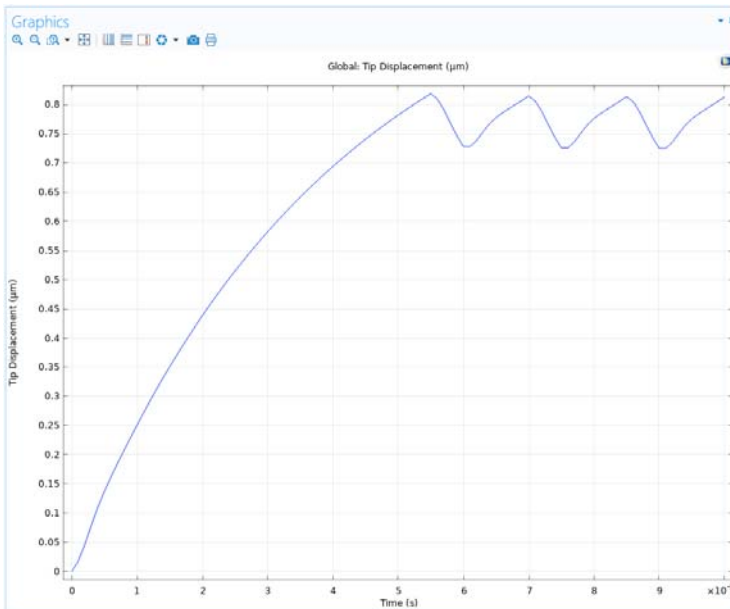
2 At the MATLAB prompt enter `mphlaunch` to launch the COMSOL Desktop that automatically loads the model object from the COMSOL Multiphysics server.

**Note:** Any modifications that you do to the model in the COMSOL Desktop modify the model loaded onto the COMSOL Multiphysics Server, and may therefore influence the simulation with Simulink, would you decide to run it again. However, for changes of the cosimulation settings, you will need to export the cosimulation file again, and then re-load it into the COMSOL Cosimulation block.

Let's create a plot of the tip displacement.

3 In the Results toolbar, click 1D Plot Group .

- 4 Select 1D Plot group 14 node, and in the Settings window for 1D Plot Group set Dataset to Study 2/ Solution 2 (sol2), which is the time dependent study used for the cosimulation.
- 5 In the Legend section, clear Show legends.
- 6 In the 1D Plot Group 14 toolbar, click Global .
- 7 In the Settings window for Global, locate the y-Axis Data section. In the Expression column of the table enter `-comp1.pppb1`. In the Description column enter Tip Displacement.
- 8 Click Plot .



Notice the difference between this plot and the plot displayed in Simulink. According to the cosimulation setup, in the COMSOL model the solution is stored at the output time steps of the COMSOL solver (every 1  $\mu$ s).

**Note:** If you plan to run a new Simulink simulation with the current model, it is important that you close the COMSOL Desktop first.