



# INTRODUCTION TO LiveLink™ for Excel®

# Introduction to LiveLink™ for Excel®

© 2012–2020 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. Microsoft, Excel, Visual Basic and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States and/or other countries. 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: CM023402

# Contents

---

Introduction .....	5
Working with COMSOL® Models in Excel® .....	6
Exporting Material Data .....	24
Loading and Saving Excel® Files from the COMSOL Desktop® .....	36



## Introduction

---

LiveLink™ *for* Excel® allows you to take advantage of the capabilities and structured simplicity offered by Microsoft® Excel® to extend your COMSOL Multiphysics® modeling capacity. Parameters and variables that are defined and modeled in COMSOL Multiphysics are instantly available in Excel and automatically synchronized with your physics model.

Excel® files can be loaded and saved from COMSOL Desktop. This can be used for handling parameters, variables, interpolation data etc. as well as loading and saving Excel files from applications.

LiveLink™ *for* Excel® adds the capability to create a COMSOL® material library from data stored in a worksheet and brings it into COMSOL Multiphysics.

**Note:** Connecting Excel to a COMSOL Multiphysics Server is only supported on Windows®. Loading and saving files is supported on all platforms and doesn't require Excel to be installed.

# Working with COMSOL® Models in Excel®

---

LiveLink™ for Excel® enables direct access of model definitions such as parameters, variables, and functions by using the tools from the COMSOL ribbon tabs in Excel. You can also modify and update the model, recompute the solution, and extract results to a worksheet. The step-by-step instructions below detail how to work with a model in Excel spreadsheets, covering the stages of:

- Opening and saving COMSOL models
- Retrieving and updating Model Parameters
- Computing the solution and extracting results
- Updating existing data in a linked workbook

The example is based on a modified version of the model, Electrical Heating in a Busbar, from the COMSOL Multiphysics application library. The model analyzes the resistive heating of a busbar designed to conduct direct current; for details, see the *Introduction to COMSOL Multiphysics*.

**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 not being available for the following sections. Start with [Opening a Model](#) and work through the sections until reaching the last section, [Opening a Worksheet Linked to a Model](#).

## Opening a Model

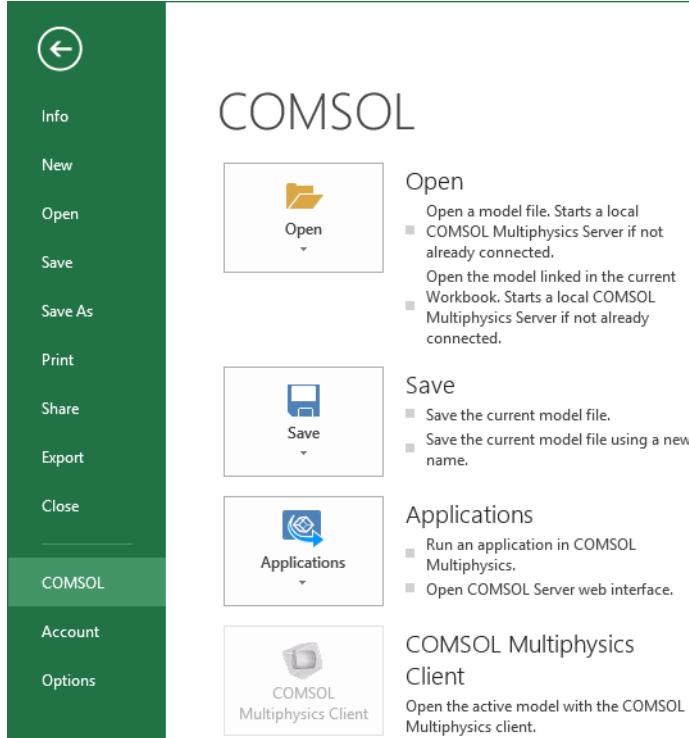
---

- 1 Start Excel.
- 2 The first time you start Excel after having installed COMSOL you may be met with a dialog asking permission to install the COMSOL Add-in. Click OK to do so. The first result of a successful installation is that the COMSOL 5.6 Ribbon tab appears in Excel.



After this you can load a COMSOL model.

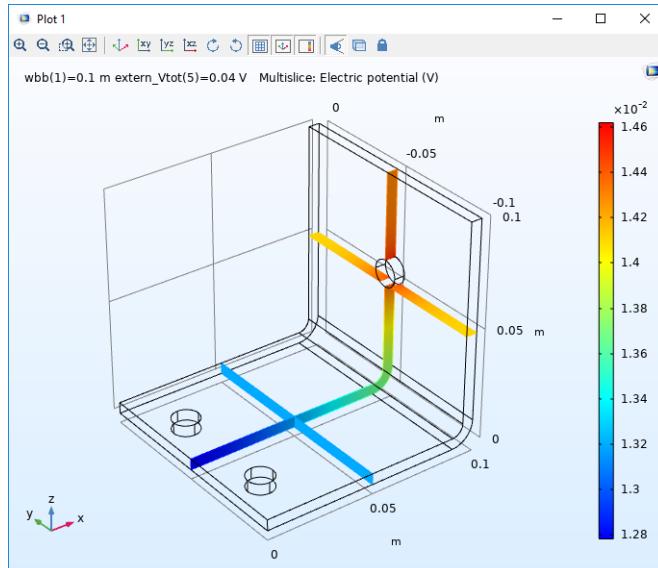
3 In the Excel File menu, select COMSOL to open the COMSOL backstage view.



**Note:** In the COMSOL backstage view you can manage the COMSOL Multiphysics model, launch COMSOL Apps, connect manually to a server and set the preferences.

- 4 In the COMSOL backstage view click Open (📁).
- 5 In your COMSOL installation directory, find the folder `applications/LiveLink_for_Excel/Tutorials`.
- 6 Select the file `busbar_11excel.mph` and click the Open button.

The model file is loaded onto a COMSOL Multiphysics server, which is automatically started. A graphics window automatically displays the first plot group available in the model.



**Note:** In case no solution is computed, the graphics window shows the model geometry instead.

In the worksheet, the model path is automatically entered in cell B2.

A	B	C	D	E	F	G	H	I
1	COMSOL Model							
2	Filename:	Multiphysics\applications\LiveLink_for_Excel\Tutorials\busbar_llexcel.mph						

The cell A1 contains a comment represented by a red triangle at the upper-right corner of the cell. This comment links the workbook to the model file even if you save and reopen the workbook. How to open a linked model is described in the last section of this tutorial, [Opening a Worksheet Linked to a Model](#). Do not remove this comment unless you want to break the link between the files.

A	B
1	COMSOL Model
2	Filename: Multiphysics\

**Note:** A workbook can be linked to only one COMSOL model at a time.

## Retrieving and Updating Model Parameters

---

Extract selected parameters from the model that you have just opened to a worksheet.

- 1 Select cell A4.
- 2 From the COMSOL 5.6 tab, Definitions group, click Parameters  .

This imports all the model parameters to the worksheet and creates a direct link to the Parameters nodes in the COMSOL model.

A	B	C	D	E	F	G	H
1	COMSOL Model						
2	Filename:	\Multiphysics\applications\LiveLink_for_Excel\Tutorials\busbar_llexcel.mph					
4	Parameter	Expressions	Unit	Description	Value	Unit	
5	rad_1		6 mm	Radius of bolts	0.006	m	
6	tbb		5 mm	Thickness of busbar	0.005	m	
7	wbb		10 cm	Width of busbar	0.1	m	
8	mh		6 mm	Mesh control	0.006	m	
9	Vtot		20 mV	Voltage	0.02	V	
10	L		9 cm	Length of busbar	0.09	m	

The link between the cell range in the worksheet and the COMSOL model is represented by the comment in the cell A4.

In this example we only need to access the parameter L (the length of the busbar) that is defined in the node Parameters 2.

- 3 To replace the imported parameters list by the one defined in Parameters 2 only, select cell A4. From the COMSOL 5.6 tab, Definitions group, under Parameters select Parameters 2.
- 4 Click OK in the dialog box to replace the contents of the destination cell.

A	B	C	D	E	F	G	H
1	COMSOL Model						
2	Filename:	\Multiphysics\applications\LiveLink_for_Excel\Tutorials\busbar_llexcel.mph					
4	Parameter	Expressions	Unit	Description	Value	Unit	
5	Parameters 2						
6	L		9 cm	Length of busbar	0.09	m	

**Note:** You may retrieve parameters to several locations in a worksheet or workbook. Each time you extract model parameters, a link is created for that cell range, so that you can update the model with the selected parameter list.

You may modify the parameters in the worksheet, for example by changing their expression or description. You can also add new parameters to the list. Any change

to the list is transferred to the COMSOL model during the update operation, which is by default done automatically.

You will now change the value of the L parameter to 5 cm:

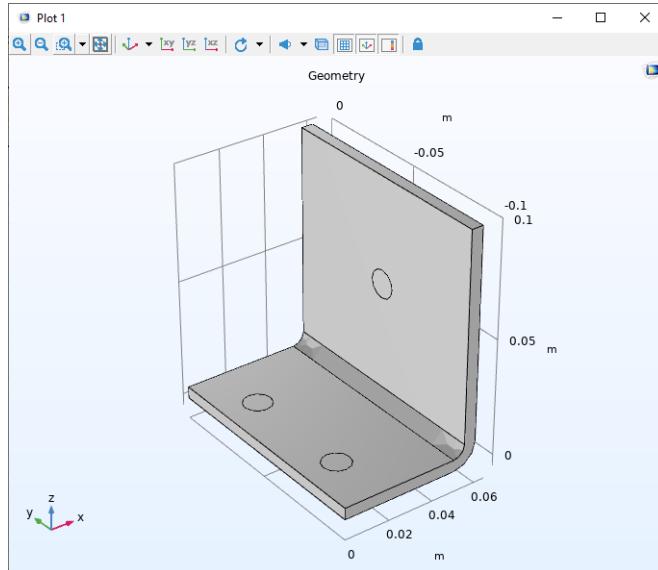
- 5 In cell B6 enter 5, the parameter L is then automatically updated to 5 in COMSOL Multiphysics.

**Note:** If you prefer a manual parameter update, go to the COMSOL preferences in the Backstage view and clear Automatic update check box. Then you can use the update button  to update the Parameters node linked to the selected cell by the comment. You can update several model definitions, such as parameters, variables, and functions, contained in the same worksheet with the Update All button  located under Update. When several Parameter table are available in the worksheet, click Update Multiple , located under Update, to select which parameter to update with the other model definitions.

To visualize the change in the model geometry due to the edited length parameter, you can display the updated geometry:

- 6 In the View group, click the Geometry button .

The graphics window should now display the geometry as shown in the figure below:



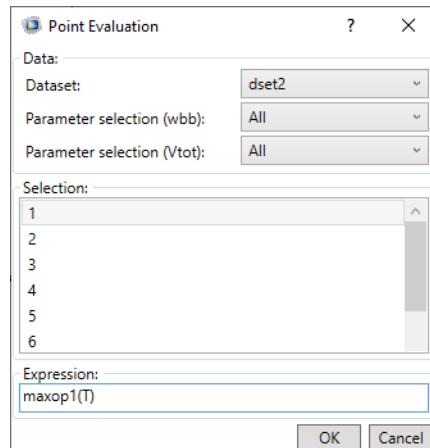
- 7 You will now set the value of parameter L back to 9 cm, in cell B6 enter 9 and press Enter.

## Extracting Results

The model loaded on the COMSOL Multiphysics server already contains a solution. It also contains an operator to evaluate the maximum in the busbar domain. Continue with the steps below to import the maximum temperature of the busbar to the worksheet.

### POINT EVALUATION

- 1 In Sheet1, select cell H4.
- 2 From the Numerical Results group, click the Point Evaluation button ().
- 3 In the dialog box that opens, select dset2 in the Dataset list. The solution dataset dset2 contains the solution stored by the parametric solver.
- 4 Select point 1 from the Selection list and enter `maxop1(T)` in the Expression text field; then click OK.



In the Excel spreadsheet, you can now edit the cell containing the maximum temperature for several applied voltage values and a geometry width of 10 cm.

A	B	C	D	E	F	G	H	I	J
1	COMSOL Model								
2	Filename:								
3									
4	Parameter	Expressions	Unit	Description	Value	Unit			
5	Parameters 2								
6	L	5 cm	Length of busbar		0.05	m			
7									
8									
9									

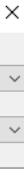
wbb (m)	Vtot (mV)	Maximum 1 (K), Point: 1
0.1	5	294.0777979
0.1	10	296.3628174
0.1	20	303.1653871
0.1	30	313.7692235
0.1	40	327.4483537

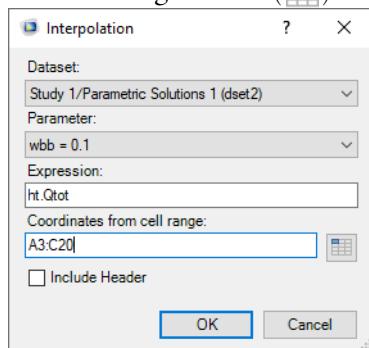
## INTERPOLATION

Continue by calculating how much heat is produced in the busbar at coordinates that you specify in the worksheet.

- 1 Add a new sheet.
- 2 In the worksheet Sheet2, enter the coordinates as shown in the figure to the right.
- 3 Select cell D3, then go to the Results group and click Interpolation .

	A	B	C
1	x	y	z
2	0	0	0
3	2.50E-02	0	0
4	5.00E-02	0	0
5	0	-1.25E-02	0
6	2.50E-02	-1.25E-02	0
7	5.00E-02	-1.25E-02	0
8	0	-2.50E-02	0
9	2.50E-02	-2.50E-02	0
10	5.00E-02	-2.50E-02	0
11	0	0	5.00E-03
12	2.50E-02	0	5.00E-03
13	5.00E-02	0	5.00E-03
14	0	-1.25E-02	5.00E-03
15	2.50E-02	-1.25E-02	5.00E-03
16	5.00E-02	-1.25E-02	5.00E-03
17	0	-2.50E-02	5.00E-03
18	2.50E-02	-2.50E-02	5.00E-03
19	5.00E-02	-2.50E-02	5.00E-03
20	0	-2.50E-02	5.00E-03

- 4 In the Dataset menu list, select Study 1/Parametric Solutions 1(dset2).
- 5 In the Interpolation dialog box, in the Expression text field, enter `ht.Qtot`, which is the total heat source to be evaluated.
- 6 In the Coordinates from cell range text field, enter `A3:C20`. This is the range containing the coordinates of the interpolation points. You can also click the Select Range button  to select the cell range.



7 Finally click OK.

	A	B	C	D	E	F	G	H
1								
2	x	y	z	0	0.000285311	0.001141244	0.004564974	0.010271192
3	0	0	0	0	82.17991479	328.7196592	1314.878637	2958.476933
4	2.50E-02	0	0	0	82.17991479	328.7196592	1314.878637	5259.514547
5	2.50E-02	0	0	0	18.24537837	72.98151349	291.926054	656.8336214
6	0	-1.25E-02	0	0	118.285023	473.1400919	1892.560368	4258.260827
7	2.50E-02	-1.25E-02	0	0	118.285023	473.1400919	1892.560368	7570.24147
8	2.50E-02	-1.25E-02	0	0	0.010089995	0.040359981	0.161439926	0.363239833
9	0	-2.50E-02	0	0	200.9570913	803.828365	3215.31346	7234.455285
10	2.50E-02	-2.50E-02	0	0	200.9570913	803.828365	3215.31346	12861.25384
11	2.50E-02	-2.50E-02	0	0	0.000237916	0.000951663	0.003806651	0.008564965
12	0	0	5.00E-03	0	81.58712785	326.3485114	1305.394046	2937.136603
13	2.50E-02	0	5.00E-03	0	81.58712785	326.3485114	1305.394046	5221.576183
14	2.50E-02	0	5.00E-03	0	17.63737522	70.5495009	282.1980036	634.9455081
15	0	-1.25E-02	5.00E-03	0	118.3704611	473.4818444	1893.927378	4261.3366
16	2.50E-02	-1.25E-02	5.00E-03	0	118.3704611	473.4818444	1893.927378	7575.709511
17	2.50E-02	-1.25E-02	5.00E-03	0	0.047698638	0.190794551	0.763178203	1.717150957
18	0	-2.50E-02	5.00E-03	0	154.6583794	618.6335177	2474.534071	5567.701659
19	2.50E-02	-2.50E-02	5.00E-03	0	154.6583794	618.6335177	2474.534071	9898.136283
20	2.50E-02	-2.50E-02	5.00E-03	0				

The evaluation results are stored in a cell range of size 18 x 5, where 18 equals the number of interpolation points and 5 is the number of parameter values contained in the solution. In this case, the solution consists of a parameter sweep over the applied voltage on the device, which varies from 5 mV to 40 mV.

8 You can format the worksheet for instance include column labels, insert cell at your convenience. Make sure that the cell comment remain in the worksheet to allow future update.

1	A	B	C	Qtot [W]				
				Vtot = 5 mV	Vtot = 10 mV	Vtot = 20 mV	Vtot = 30 mV	Vtot = 40 mV
2	x	y	z	0	0.000285311	0.001141244	0.004564974	0.010271192
3	0	0	0	0	82.17991479	328.7196592	1314.878637	2958.476933
4	2.50E-02	0	0	0	82.17991479	328.7196592	1314.878637	2958.476933
5	2.50E-02	0	0	0	18.24537837	72.98151349	291.926054	656.8336214
6	0	-1.25E-02	0	0	118.285023	473.1400919	1892.560368	4258.260827
7	2.50E-02	-1.25E-02	0	0	118.285023	473.1400919	1892.560368	4258.260827
8	2.50E-02	-1.25E-02	0	0	0.010089995	0.040359981	0.161439926	0.363239833
9	0	-2.50E-02	0	0	200.9570913	803.828365	3215.31346	7234.455285
10	2.50E-02	-2.50E-02	0	0	200.9570913	803.828365	3215.31346	7234.455285
11	2.50E-02	-2.50E-02	0	0	118.3704611	473.4818444	1893.927378	4261.3366
12	0	0	5.00E-03	0	0.000237916	0.000951663	0.003806651	0.008564965
13	2.50E-02	0	5.00E-03	0	81.58712785	326.3485114	1305.394046	2937.136603
14	2.50E-02	0	5.00E-03	0	81.58712785	326.3485114	1305.394046	2937.136603
15	0	-1.25E-02	5.00E-03	0	17.63737522	70.5495009	282.1980036	634.9455081
16	2.50E-02	-1.25E-02	5.00E-03	0	118.3704611	473.4818444	1893.927378	4261.3366
17	2.50E-02	-1.25E-02	5.00E-03	0	118.3704611	473.4818444	1893.927378	4261.3366
18	0	-2.50E-02	5.00E-03	0	0.047698638	0.190794551	0.763178203	1.717150957
19	2.50E-02	-2.50E-02	5.00E-03	0	154.6583794	618.6335177	2474.534071	5567.701659
20	2.50E-02	-2.50E-02	5.00E-03	0	154.6583794	618.6335177	2474.534071	5567.701659

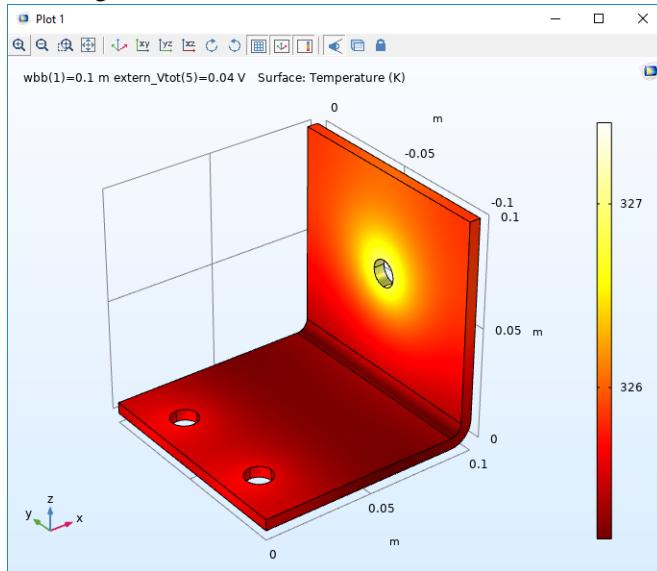
## Displaying the Solution

You can display the solution using the plot group defined in the model. You can find the plot groups available in the Plot Group list.

| Click the Plot Group button , from the Plots group, to select the plot group to display. Here, select Temperature (ht) (pg2).



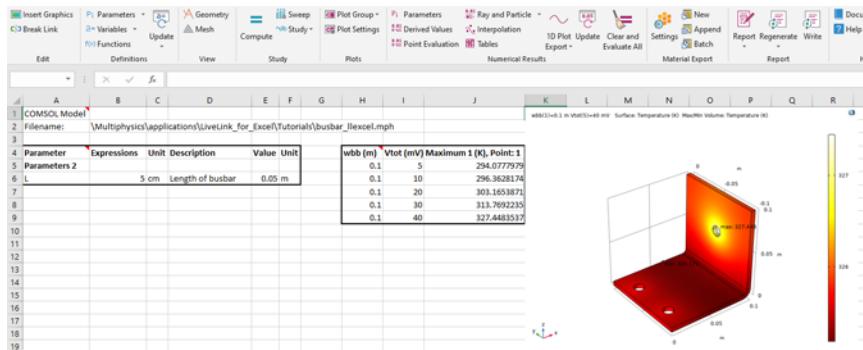
The graphics window now displays the temperature distribution in the busbar as in the figure below:



The plot represents the solution computed before the change of the busbar width.

**Note:** For time dependent solutions or parametric sweeps, you can select the solution to display by clicking the Plot Settings  button.

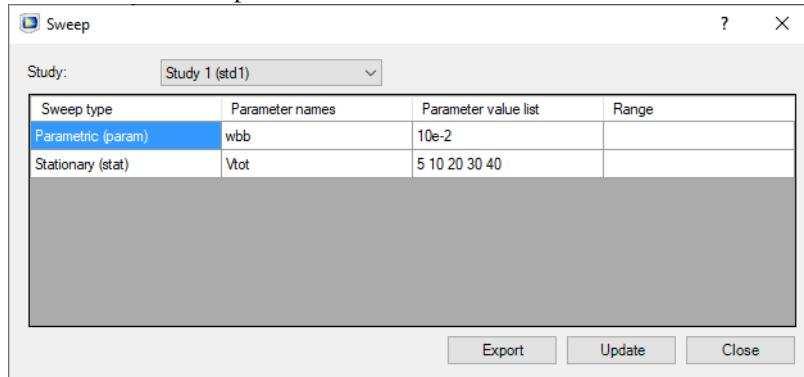
To insert the displayed image into the worksheet, in Sheet 1 select cell K1, then click Insert Graphics  from the Edit group.



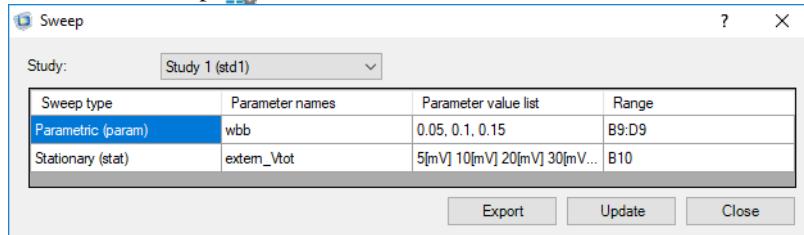
## Running Model in Sweep

The current model is set with a Parametric Sweep node and a Stationary node including a Auxiliary sweep. You can import the sweep settings and control the parameter value from the worksheet.

- 1 Select cell A8 in Sheet 1 and click Sweep  to open the Sweep dialog box. In the Sweep dialog box, you can see that the current sweep node is defined with the busbar width parameter set to the value 10e-2.



- 2 Click Export to export the sweep parameter data to the worksheet and create a link between the selected cell range and the model.
- 3 To be able to edit the worksheet, first click Close to close the Sweep dialog box.
- 4 Add additional parameter values to the parametric sweep, in cell B9 enter  $5e-2$ , in C9 enter 0.1 and in D9 enter 0.15.
- 5 You will now update the model with the additional parameter values, select A8 and click Sweep .



- 6 You may notice that the parameter value list and the range columns are updated with the new values. Click Update to send these parameter values to the model.

## Computing the Solution

---

Continue with solving the model.

- In the Study group, click the Compute button .

For the case where you have several studies in your model, you can click Study  to select which study to solve.

## Updating Data in a Worksheet

---

Now that you have a workbook containing model definitions and data evaluation linked to a COMSOL model, you can easily modify and recompute the model from the workbook, while keeping the evaluated results up to date.

Follow the steps below to update the results in Sheet1 and Sheet2.

- In the Numerical Results group, click Update  to update all the numerical results in the worksheet.

The point evaluation now includes results for different values of the busbar width.

- To update the imported image, you need to delete the existing one and insert it again by clicking Insert Graphics. Select Plot Settings  to display the solution for each value of the busbar width.

A	B	C	D	E	F	G	H	I	J
1 COMSOL Model									
2 Filename:									
3									
4 Parameter	Expressions	Unit	Description	Value	Unit				
5 Parameters 2									
6 L	5 cm	Length of busbar		0.05	m				
7									
8 Name	Expressions								
9 wbb	5.00E-02	0.1		0.15					
10 Vtot	5	10		20	30	40			
11									
12									
13									
14									
15									
16									
17									
18									
19									

wbb (m)	Vtot (mV)	Maximum 1 (K), Point: 1
0.05	5	294.7634786
0.05	10	298.3921785
0.05	20	310.0511019
0.05	30	327.7515748
0.05	40	350.2577645
0.1	5	294.0777979
0.1	10	296.3628174
0.1	20	303.1653871
0.1	30	313.7692235
0.1	40	327.4483537
0.15	5	293.8077613
0.15	10	295.5113275
0.15	20	300.6519783
0.15	30	308.2180048
0.15	40	318.360272

3 To update the total heat source interpolation only, switch to Sheet2, and select the cell that contains the Interpolation comment, cell D3 if you did not change the worksheet format.

4 In the Numerical Results group click Interpolation .

5 The Interpolation dialog box already contains the appropriate settings for the evaluation. Click OK again in the dialog box.

	A	B	C	D	E	F	G	H
1				Qtot [W]				
2	x	y	z	Vtot = 5 mV	Vtot = 10 mV	Vtot = 20 mV	Vtot = 30 mV	Vtot = 40 mV
3	0	0	0	0.117269374	0.469077494	1.876309976	4.221697447	7.505239905
4	2.50E-02	0	0	0.477.5731914	1910.292766	7641.171063	17192.63489	30564.68425
5	2.50E-02	0	0	0.477.5731914	1910.292766	7641.171063	17192.63489	30564.68425
6	0	-1.25E-02	0	0.007954612	0.031818448	0.127273791	0.28636603	0.509095164
7	2.50E-02	-1.25E-02	0	0.433.0235313	1732.094125	6928.376502	15588.84713	27713.50601
8	2.50E-02	-1.25E-02	0	0.433.0235313	1732.094125	6928.376502	15588.84713	27713.50601
9	0	-2.50E-02	0	0.019301638	0.077206553	0.308826212	0.694858977	1.235304847
10	2.50E-02	-2.50E-02	0	0.478.8569834	1915.427934	7661.711735	17238.8514	30646.84694
11	2.50E-02	-2.50E-02	0	0.478.8569834	1915.427934	7661.711735	17238.8514	30646.84694
12	0	0	5.00E-03	0.168721321	0.674885283	2.699541133	6.07396755	10.79816453
13	2.50E-02	0	5.00E-03	0.478.4777036	1913.910814	7655.643257	17225.19733	30622.57303
14	2.50E-02	0	5.00E-03	0.478.4777036	1913.910814	7655.643257	17225.19733	30622.57303
15	0	-1.25E-02	5.00E-03	0.026275896	0.105103583	0.420414333	0.945932249	1.681657331
16	2.50E-02	-1.25E-02	5.00E-03	363.9985066	1455.994026	5823.976106	13103.94624	23295.90442
17	2.50E-02	-1.25E-02	5.00E-03	363.9985066	1455.994026	5823.976106	13103.94624	23295.90442
18	0	-2.50E-02	5.00E-03	0.016769615	0.067078461	0.268313844	0.603706148	1.073255374
19	2.50E-02	-2.50E-02	5.00E-03	477.8078349	1911.23134	7644.925359	17201.08206	30579.70144
20	2.50E-02	-2.50E-02	5.00E-03	477.8078349	1911.23134	7644.925359	17201.08206	30579.70144

Note that the new value correspond to the busbar width set to 5 cm. You need to update the formatting manually.

6 You can continue to import data for other value of wbb, the easiest way is to copy the cell that contains the interpolation comment (D3) and paste it at the desired cell location, say I3. Then click Interpolation button and select in the Parameter menu list the second parameter (wbb = 0.1) and click OK. A

COMSOL dialog box pops up, click OK to confirm the change of the contents of the destination cells.

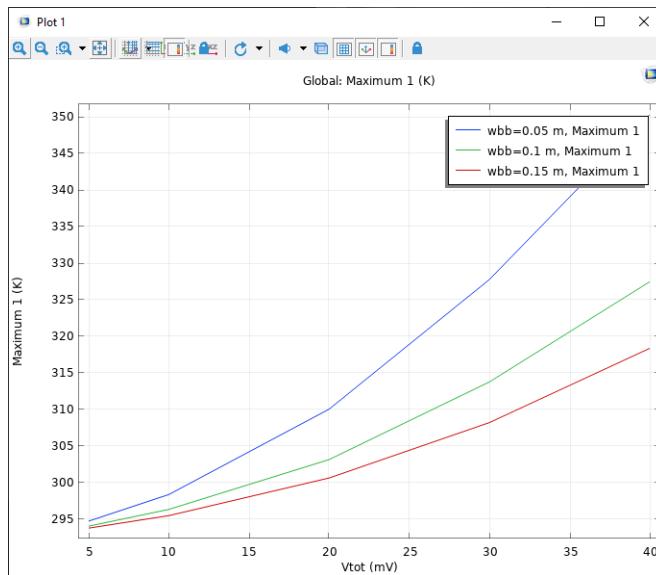
1	A	B	C	D	E	F	G	H	I	J	K	L	M
2	x	y	z	Vtot = 5mV	Vtot = 10mV	Vtot = 20mV	Vtot = 30mV	Vtot = 40mV	Vtot = 5mV	Vtot = 10mV	Vtot = 20mV	Vtot = 30mV	Vtot = 40mV
3	0	0	0	0.121610671	0.486442686	1.945707743	4.377884171	7.78308297	0.000273769	0.001050576	0.004380304	0.009855684	0.017521217
4	2.50E-02	0	0	0.478.0639838	1912.255935	7649.02374	21710.30342	30596.09496	82.38715782	329.5496313	1318.194525	2965.937681	5272.7781
5	5.00E-02	0	0	451.713616	1805.854464	7227.417856	16261.69018	28909.67142	117.8517546	471.4070184	1885.628074	4242.653165	7542.512295
6	0	-1.25E-02	0	0.0085497	0.034198801	0.136795203	0.307789208	0.547180811	18.34902588	73.39610352	293.5844141	660.5649317	1174.337656
7	2.50E-02	-1.25E-02	0	453.7681952	1813.072781	7260.291123	16355.65505	29041.16449	118.9871057	475.9484229	1903.739392	4283.335806	7615.174767
8	5.00E-02	-1.25E-02	0	451.6142856	1806.457142	7225.02857	16258.11428	28903.31428	119.9985817	479.9943267	1919.977307	4319.94694	7679.909227
9	0	-2.50E-02	0	0.01791893	0.071675973	0.286703893	0.64508376	1.146815574	0.012145696	0.048582784	0.194331135	0.437245055	0.777324547
10	2.50E-02	-2.50E-02	0	480.141548	1920.566192	7682.204768	17285.09573	30729.05907	206.9540036	827.8160146	3311.264058	7450.344131	13245.05623
11	5.00E-02	-2.50E-02	0	451.8155935	1807.262374	7229.049496	16265.36137	28916.19799	122.6022307	490.408927	1961.635692	4413.680307	7846.542767
12	0	0	5.00E-03	0.160269988	0.641079953	2.564319812	5.769719577	10.25727925	0.000226179	0.000904718	0.003618871	0.00814240	0.014475485
13	2.50E-02	0	5.00E-03	479.0510267	1916.204107	7664.816427	17245.83694	30659.26571	81.80950314	327.2380125	1308.95205	2945.142113	5235.808201
14	5.00E-02	0	5.00E-03	451.7157371	1808.862948	7227.451793	16261.76653	28809.80717	117.8220288	471.2881151	1885.15246	4241.593036	7540.609841
15	0	-1.25E-02	5.00E-03	0.027903349	0.111613397	0.446453586	1.004520569	1.785814345	17.81110244	71.24440974	284.977639	641.1996877	1139.910556
16	2.50E-02	-1.25E-02	5.00E-03	363.9640818	1455.856327	5823.425309	13102.70694	23293.70124	118.3421426	473.3685704	1893.474282	4260.317133	7573.897126
17	5.00E-02	-1.25E-02	5.00E-03	451.6073387	1806.429355	7225.717419	16257.86419	28902.86967	119.9965884	479.9883534	1919.945414	4319.877181	7679.781654
18	0	-2.50E-02	5.00E-03	0.015632227	0.062528909	0.250115637	0.562760184	1.000462549	0.027591537	0.11036615	0.441464598	0.993295347	1.765883894
19	2.50E-02	-2.50E-02	5.00E-03	477.969502	1911.878008	7647.512032	17206.90207	30598.04813	153.2903106	613.1612433	2452.644973	5518.45119	9810.579803
20	5.00E-02	-2.50E-02	5.00E-03	451.80337029	1807.214811	7228.859246	16264.93333	28915.43698	122.8034548	490.4138192	1961.655277	4413.74373	7846.621107

7 Finally repeat the step above to import the results using the third parameter (wbb = 0.15).

## Importing 1D Plot Data

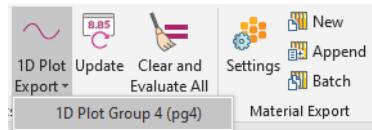
Excel does not support general 2D- and 3D-plots that are commonly produced in COMSOL. However, 1D-plots have data that can be extracted to Excel and Excel can create plots of these data as well.

In the busbar\_llexcel model there is a single 1D-plot that looks like this when plotted using Excel:

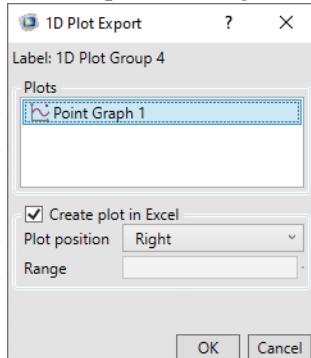


This plot can be inserted into Excel as a static image. Instead we will insert the data that makes up the plot and create a scatter chart in the Excel sheet.

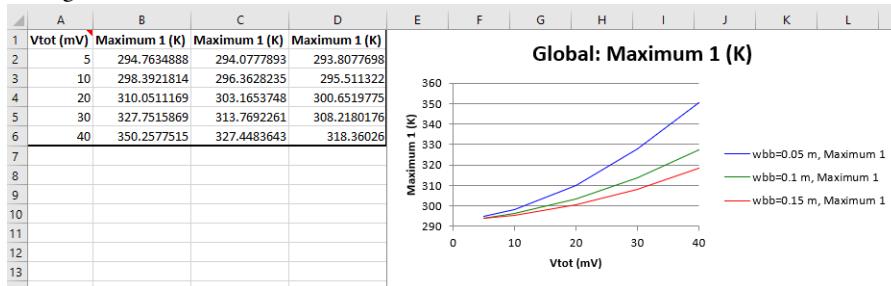
- 1 Add a new sheet.
- 2 Select cell A1 in Sheet 3, and click the 1D Plot Export button from the Plots group, to select the plot group to display. Here, select 1D Plot Group 4 (pg4).
- 3 On the COMSOL ribbon tab, locate the 1D Plot Export button. Click on it see what 1D plots are available in the model and select 1D Plot Group 4 (pg4).



4 This opens a dialog box when you can set the data to be imported into Excel.



Here it is possible to adjust what plot features should be imported in case the 1D-plot contains more than one plot. It is possible to control if a plot should be generated in addition to imported the numerical data. Note that the numerical data have to be imported in order for Excel to be able to generate a chart. Use the plot position to select where the chart should be placed relative to the imported data. For most plots it works best to select **right** or **bottom**. It is possible to select **custom** and choose a specific location. The final chart looks like this when using the default settings:

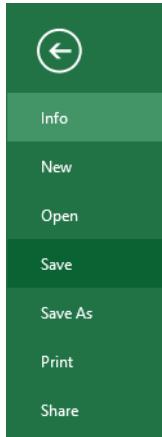


## Saving the Model and the Workbook

To keep the changes you have applied to the busbar model, save it to the MPH-format. To avoid writing over the model from the Application Libraries, use the Save As button.

| Go to the Excel File menu and select COMSOL. In the COMSOL backstage view click Save button and select Save As , then save the file to a location outside the COMSOL Application Libraries.

2 Continue with saving the Excel workbook, go to the File menu and select Save.



3 You can now exit Excel, which automatically closes the COMSOL Model window as well.

## Opening a Worksheet Linked to a Model

---

Once you have created a link to a COMSOL model in an Excel workbook and saved the file, you can reopen the Excel file and update the link.

- 1 Start Excel and open the file you have saved in step 2 of the section [Saving the Model and the Workbook](#).
- 2 Go to the Excel File menu and select COMSOL. In the COMSOL backstage view click Open button  and select Open linked .

**Note:** The path of the linked model is set in the COMSOL Model cell comment. If necessary you can manually edit the cell comment to update the model path.

You can now continue your work with the model in Excel.

## Automation Using VBA

---

Using Visual Basic for Application (VBA) you can write macros in Excel worksheet that can automate operations between the worksheet and the COMSOL model. For instance it is possible to generate a macro that does the operation above automatically: update the sweep parameter, compute the solution and update the results in the worksheet.

For more information please refer to LiveLink for Excel User's Guide Manual. You can also open the file `busbar_11excel.xlsxm` that you can find in your COMSOL Multiphysics installation directory and in the folder `applications/LiveLink_for_Excel/Tutorials`

# Exporting Material Data

---

Using LiveLink™ for Excel® you can easily convert material properties saved in a worksheet to a COMSOL material library. The Material Export group of the COMSOL tab in Excel contains the tools to format and export the material data. Follow the instructions below to create a user-defined material library that will be available automatically in the Material Browser in the COMSOL Desktop.

In this section, step-by-step instructions show you how to export the data stored in a spreadsheet to a material library in the COMSOL format. The first part of the example shows how to define the export settings from constant data. The second part of the example illustrates how to set the export from data field depending on the physical quantity; you will consider temperature in this particular example. Finally, in the last part, you will export the data from the spreadsheet to a COMSOL material library.

## Defining Material Export Settings from Constant Data

---

In this section, you will set up the material export using data stored as constants in the spreadsheet.

**Note:** The export procedure described in this chapter assumes that the data are stored in the spreadsheet with the material names in a single column and the material property names in a single row. The data are placed at the intersection cell of the material name and the property name.

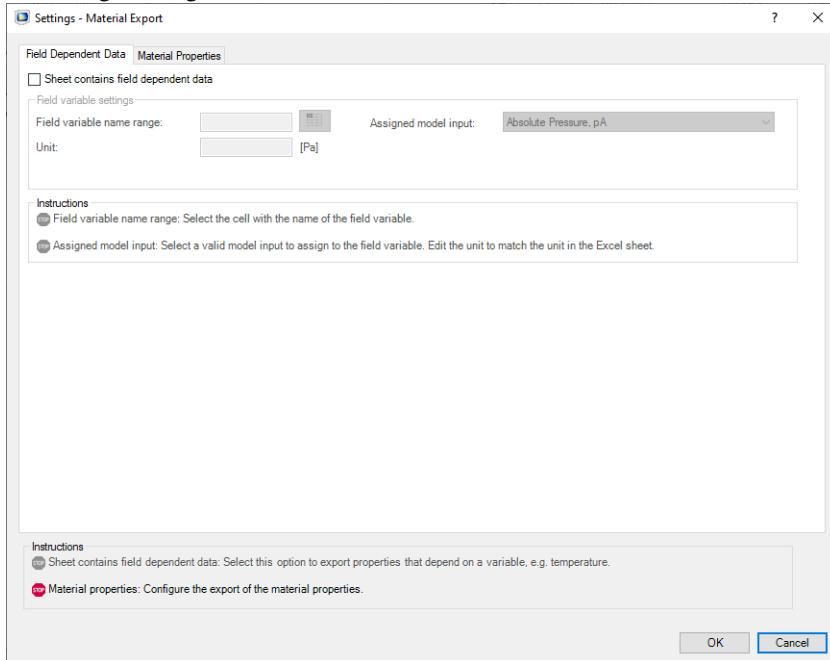
	A	B	C
1		thermal conductivity (W/m/K)	
2	Copper	400	3.14E-01
3	Titanium	10	1.79E-01
4	Aluminum	150	9.75E-02

- 1 Start Excel and open the file `busbar_11excel_data.xlsx` that you can find in your COMSOL Multiphysics installation directory and in the folder `applications/LiveLink_for_Excel/Tutorials`.
- 2 Go to the worksheet **Materials** which contains the definitions of physical properties for four different materials. The material properties listed are thermal conductivity, density, heat capacity at constant pressure, relative permittivity and

electric conductivity. All properties are defined in SI units, except for density, which has the unit lb/in<sup>3</sup>.

A	B	C	D	E	F
1	thermal conductivity (W/m/K)	density (lb/in <sup>3</sup> )	Cp (J/kg/K)	Relative permittivity	sigma (S/m)
2	Copper	400	3.14E-01	380	1 6.00E+07
3	Titanium	10	1.79E-01	700	1 7.40E+05
4	Aluminum	150	9.75E-02	900	1 3.50E+07
5	Steel	50	2.82E-01	480	1 4.00E+06

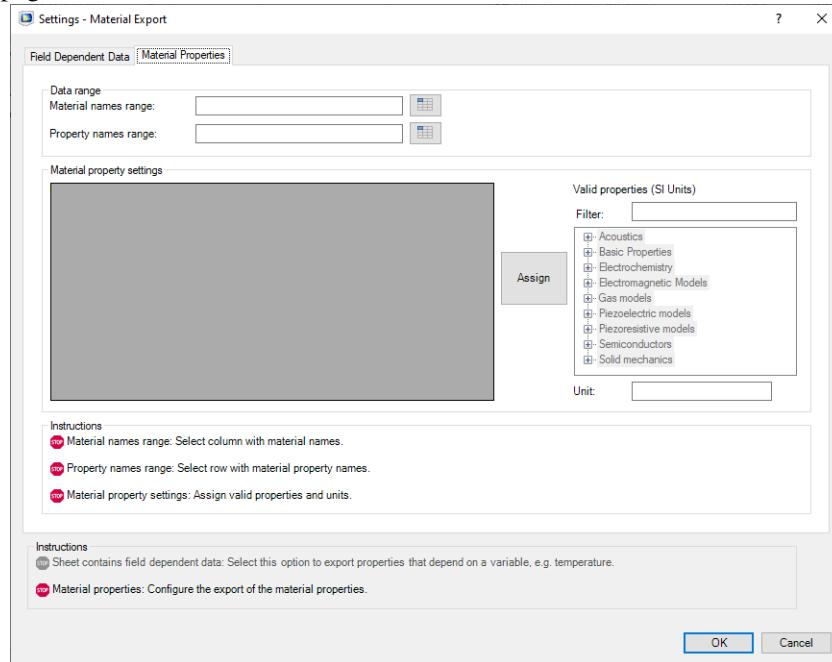
3 On the COMSOL 5.6 tab, click Settings  to open the Material Export Settings dialog box.



In the Settings dialog box you can select materials and properties to export. This is also where you associate each material property from the worksheet with a property recognized by COMSOL.

Export depends on the format of the data stored in the worksheet; you can have material properties defined with constant values or ones that depend on field data, such as temperature. In the worksheet Materials, you can see that the properties are defined as constant.

4 In the Field Dependent Data page, make sure that the Sheet contains field dependent data check box is not selected and go to the Material Properties page.

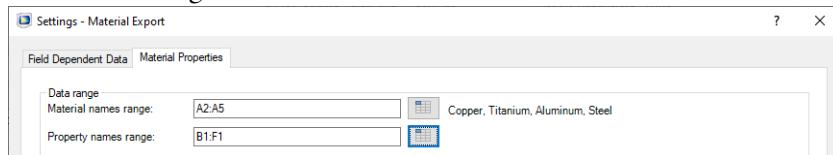


5 You will now select the cells containing the material names. Click the Range button (grid icon) next to the Material names range text field.

6 Select the range A2:A5 in the sheet and click OK.

7 Continue by selecting the cells with the property names. Click the Range button (grid icon) to the right of the Property names range text field.

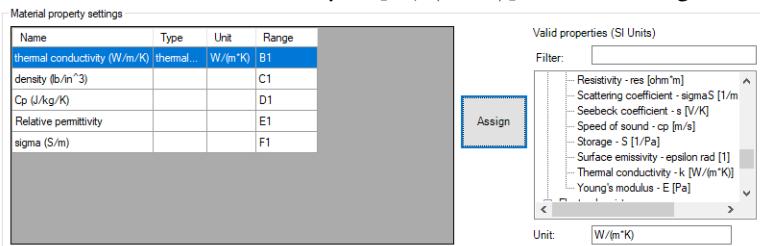
8 Select the range B1:F1 and click OK.



The properties are now listed in the table in the Material property settings section. You now need to assign valid COMSOL properties to the material properties in the spreadsheet.

9 From the Material property settings table select thermal conductivity (W/m/K), then go to the Valid properties (SI Units) list and expand Basic Properties.

**10** Select Thermal conductivity -k [W/(m\*K)] and click Assign.



Material property settings

Name	Type	Unit	Range
thermal conductivity (W/m/K)	thermal...	W/m <sup>2</sup> K	B1
density (lb/in <sup>3</sup> )			C1
Cp (J/kg/K)			D1
Relative permittivity			E1
sigma (S/m)			F1

Valid properties (SI Units)

Filter:

- Resistivity - res [ohm\*m]
- Scattering coefficient - sigmaS [1/m]
- Seebeck coefficient - s [V/K]
- Speed of sound - cp [m/s]
- Storage - S [1/Pa]
- Surface emissivity - epsilon rad [1]
- Thermal conductivity - k [W/m<sup>2</sup>K]
- Young's modulus - E [Pa]

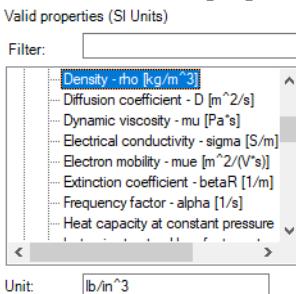
Unit:

Note that the number of properties remaining to be assigned is displayed in the Instructions section.

**11** Now select density (lb/in<sup>3</sup>) from the Name column, then select Density - rho [kg/m<sup>3</sup>] from the Valid properties (SI Units) list.

You need to specify the material properties unit if these are not defined in SI units in the worksheet.

**12** Below the valid properties list, locate the Unit text field and enter **lb/in<sup>3</sup>**.



Valid properties (SI Units)

Filter:

- Density - rho [kg/m<sup>3</sup>]
- Diffusion coefficient - D [m<sup>2</sup>/s]
- Dynamic viscosity - mu [Pa\*s]
- Electrical conductivity - sigma [S/m]
- Electron mobility - mue [m<sup>2</sup>/(V\*s)]
- Extinction coefficient - betaR [1/m]
- Frequency factor - alpha [1/s]
- Heat capacity at constant pressure

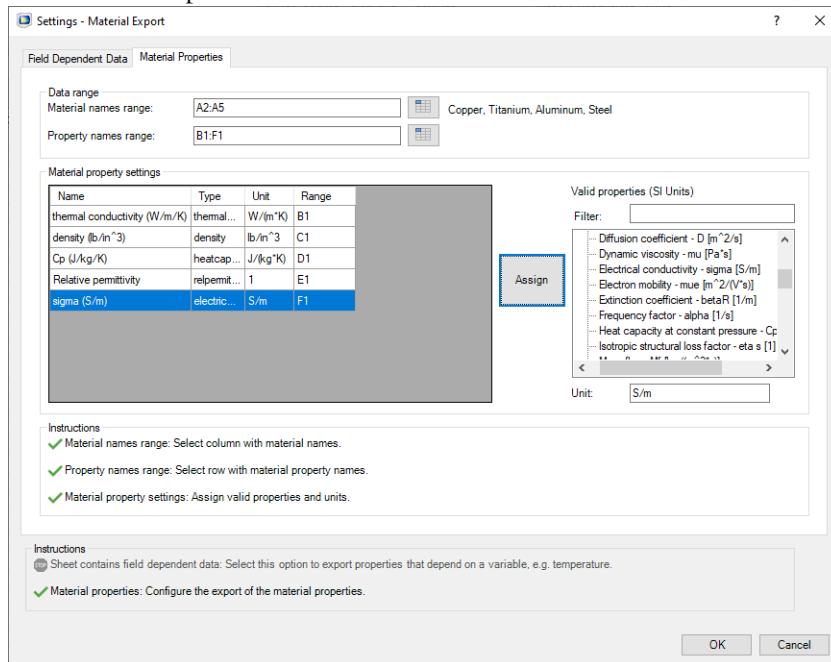
Unit:

**13** Click Assign to assign the properties and the unit to the data in the spreadsheet.

14 Repeat step 8 and 9 for the remaining material properties. Select valid properties according to the table below:

PROPERTY NAMES	VALID PROPERTIES (SI UNITS)	UNIT
Cp (J / (kg*K))	Heat capacity at constant pressure - Cp [J/(kg*K)]	J / (kg*K)
Relative permittivity	Relative permittivity - epsilon_r [1]	1
sigma (S/m)	Electrical conductivity - sigma [S/m]	S/m

Once there are no unassigned properties remaining, a check mark appears next to the last step in the Instructions section.



15 Click OK to save the settings, and to close the dialog box.

Note that comments now appear on the cells containing the material names and properties.

A	B	C	D	E	F
1	thermal conductivity (W/m/K)	density (lb/in <sup>3</sup> )	Cp (J/kg/K)	Relative permittivity	sigma (S/m)
2 Copper	400	3.14E-01	380	1	6.00E+07
3 Titanium	10	1.79E-01	700	1	7.40E+05
4 Aluminum	150	9.75E-02	900	1	3.50E+07
5 Steel	50	2.82E-01	480	1	4.00E+06

These comments contain the configuration for the material data export that you have just set up. Make sure not to remove them before completing the export. To modify the settings you can return to the Settings dialog box.

You can also use these settings as a template to run the export in a batch.

## Defining Material Export Settings from Field Data

In this section you will continue to define the material export settings, but this time using data defined as fields that depend on a variable. A valid field variable corresponds to model inputs in the COMSOL model. Typical examples include physical quantities such as temperature, concentration, and frequency. See [About Model Inputs](#) in the *COMSOL Multiphysics Reference Manual* to get a complete list of valid model inputs.

You can choose to export the material properties as tabulated data or constant values.

**Note:** The export procedure described in this chapter assumes that data is stored in the spreadsheet with the field variable data in a single column and the material property names in a single row. The data are placed at the intersection cells of the field data and the property name. It also assumes one material per sheet.

A	B
1 Water, liquid	
2 T (K)	eta (Pa*s)
3 273.15	0.0017915
4 278.15	0.0015356
5 283.15	0.0013248
6 288.15	0.0011516

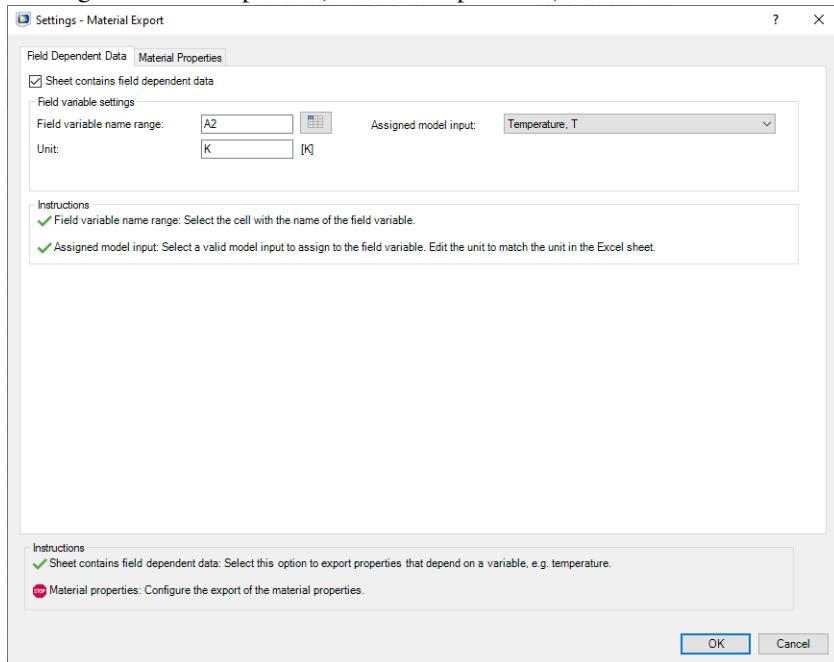
In the workbook `busbar_llexcel_data.xlsx`, go to the worksheet `Water, liquid` that contains the definitions for the dynamic viscosity, heat capacity at constant pressure, density, and thermal conductivity of liquid water. The

spreadsheet includes property data in the temperature range 273.15 K to 518.15 K.

A	B	C	D	E	
1	Water, liquid				
2	T (K)	eta (Pa*s)	Cp (J/(kg*K))	rho (kg/m^3)	k (W/(m*K))
3	273.15	0.0017915	4216.2779	1003.9209	0.55623
4	278.15	0.0015356	4206.8543	1003.0467	0.56642
5	283.15	0.0013248	4198.8844	1002.0375	0.57614
6	288.15	0.0011516	4192.27	1000.8935	0.58541
7	293.15	0.0010093	4186.9181	999.6151	0.59423
8	298.15	0.00089255	4182.7412	998.2024	0.60261
9	303.15	0.00079641	4179.6571	996.6558	0.61055
10	308.15	0.00071689	4177.5892	994.9755	0.61807
11	313.15	0.00065064	4176.4663	993.1619	0.62516
12	318.15	0.0005949	4176.2226	991.2152	0.63184
13	323.15	0.00054741	4176.7979	989.1356	0.6381
14	328.15	0.00050638	4178.1371	986.9235	0.64396
15	333.15	0.0004704	4180.1909	984.5792	0.64942

- 2 On the COMSOL 5.6 tab, click Settings  to open the Material Export Settings dialog box.
- 3 In the Field Dependent Data page, select Sheet contains field dependent data check box.
- 4 Now select the range where the field variable is defined. Click the Range button (  ) next to the Field variable name range text field and select the range A2 in the sheet. Click OK to validate the selection.

5 In the next step, assign a valid model input to the selected variable field. In the Assigned model input list, select Temperature, T.



**Note:** If the data stored in the worksheet are not defined in SI units, you need to update the Unit text field.

You can verify that all requirements in the field dependent page are set when the instructions list only contains green check marks (✓).

6 Go to the Material properties page to continue with the material library export settings.

7 You will now select the cells containing the material names. Click the Range button (grid icon) next to the Material names range text field.

8 Select cell A1 in the sheet and click OK.

9 Continue by selecting the cells with the property names. Click the Range button (grid icon) to the right of the Property names range text field.

10 Select the range B2:E2 and click OK.

The properties are now listed in the table in the Material property settings section. You now need to assign valid COMSOL properties to the material properties in the spreadsheet.

11 From the Material property settings table select eta (Pa\*s), then go to the Valid properties (SI Units) list and expand Basic Properties.

12 Select Dynamic viscosity - mu [Pa\*s] and click Assign.

Material property settings

Name	Type	Unit	Range	Constant
eta (Pa*s)	dynamic...	Pa*s	B2	<input checked="" type="checkbox"/>
Cp (J/(kg*K))			C2	<input type="checkbox"/>
rho (kg/m^3)			D2	<input type="checkbox"/>
k (W/(m*K))			E2	<input type="checkbox"/>

Valid properties (SI Units)

Filter:

Unit: Pa\*s

Instructions

- ✓ Material names range: Select column with material names.
- ✓ Property names range: Select row with material property names.
- ⚠ Material property settings: Assign valid properties and units.

Assign

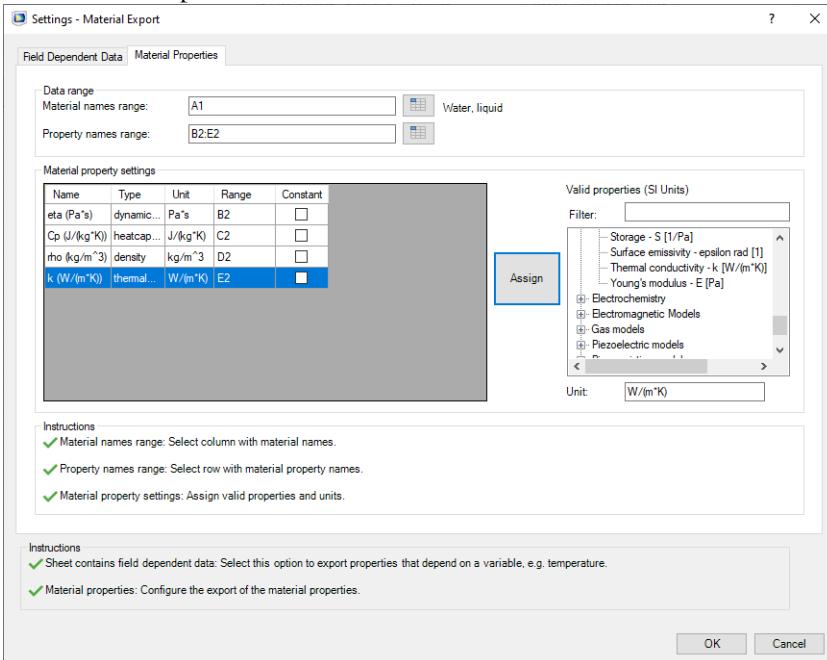
Dynamic viscosity - mu [Pa\*s]  
Electrical conductivity - sigma [S/m]  
Electron mobility - mue [m^2/(V\*s)]  
Extinction coefficient - betaR [1/m]  
Frequency factor - alpha [1/s]  
Heat capacity at constant pressure  
Isotropic structural loss factor - eta s  
Mass flux - M [kg/(m^2\*s)]

Note that the number of properties remaining to be assigned is displayed in the Instructions section.

**I3** Repeat steps **I1** and **I2** for the remaining material properties. Select valid properties according to the table below:

PROPERTY NAMES	VALID PROPERTIES (SI UNITS)	UNIT
$C_p$ (J/(kg*K))	Heat capacity at constant pressure - $C_p$ [J/(kg*K)]	J/(kg*K)
$\rho$ (kg/m <sup>3</sup> )	Density - $\rho$ [kg/m <sup>3</sup> ]	kg/m <sup>3</sup>
$k$ (W/(m*K))	Thermal conductivity - $k$ [W/(m*K)]	W/(m*K)

Once there are no unassigned properties remaining, a check mark appears next to the last step in the Instructions section.



**Note:** Select Constant dialog box to export a specific property as constant. For constant export, select the cell range of the property values in the Property name range.

**I4** Click OK to save the settings, and to close the dialog box.

15 Note that comments now appear on the cells containing the material names and properties.

	A	B	C	D	E
1	Water, liquid				
2	T (K)	eta (Pa*s)	Cp (J/(kg*K))	rho (kg/m^3)	k (W/(m*K))
3	273.15	0.0017915	4216.2779	1003.9209	0.55623
4	278.15	0.0015356	4206.8543	1003.0467	0.56642
5	283.15	0.0013248	4198.8844	1002.0375	0.57614

## Exporting the Material Properties

---

Now that you have defined the material export settings for data stored in the workbook you can finalize the export to a new material library.

- 1 Go to Materials worksheet.
- 2 To create a material library using the selected data in the spreadsheet, click  in the Material Export group. This automatically starts the COMSOL Multiphysics Server, if not already started, and the Save Material Library window.
- 3 In the Save Material Library window browse to the folder `.comsol/v56/materials` available in the user local directory.
- 4 In File name text field, enter a name for the material library. For this example, enter `My_Materials` and click Save.
- 5 Now go to the Water, liquid worksheet.
- 6 To append the material to the library created previously, click the Append () button.
- 7 In the Open Material Model window, select `My_Materials.mph` and click Open. The next time you start the COMSOL Desktop, the Material Browser will be automatically updated with the new library saved in the `.comsol/v56/materials` folder.

## Loading the Materials in the COMSOL Desktop®

---

Now that you have created a user-defined material library, you can access the material properties directly in the COMSOL Desktop.

- 1 Start the COMSOL Desktop.
- 2 Click Model Wizard .

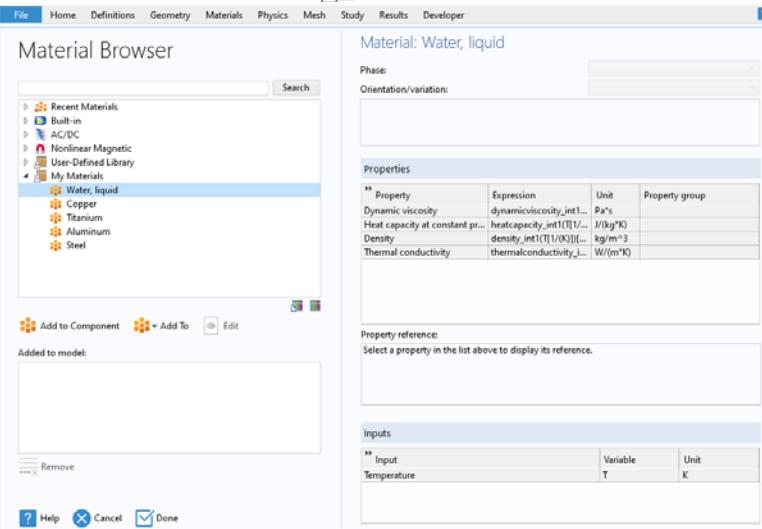
3 For the space dimension click 3D .

4 Click Done .

5 On the Materials toolbar, click Browse Materials .

**Note:** An alternative method to search for a material is to enter the material name in the search text field.

6 Expand the My Materials  node and select Water, liquid.



7 Click Add to Component  to include the material properties in the model.

8 Click Done (  ) to close the Material Browser page.

## Loading and Saving Excel® Files from the COMSOL Desktop®

---

LiveLink™ for Excel® adds the Excel XLSX format in the list of supported file formats for loading and saving data while setting up applications in the COMSOL Desktop. Feature nodes that support this functionality include Parameters, Variables, Interpolation functions, Piecewise functions, Parametric Sweep, the continuation section of the Stationary node, and the auxiliary sweep section. The data formats used by these features may differ. The easiest way to find out how to organize data in the Excel file is to enter some settings in a table in the COMSOL Desktop and save it to the Excel format.

In the current example, you will modify an existing model by importing model definitions from an Excel file. You will start by loading the model, Electrical Heating in a Busbar, from the COMSOL Multiphysics application library. This model analyzes the resistive heating of a busbar designed to conduct direct current. For details, see the booklet *Introduction to COMSOL Multiphysics*.

The tutorial walks you through how to import Excel files for three feature nodes, each with a different requirement on the data format. The feature nodes that are covered are:

- Parameters
- Interpolation function
- Continuation section of the Stationary study step node

### Opening the Model

---

- 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>busbar and click Open Application.

## Importing Parameters from Excel®

1 In the Model Builder, find and expand the Global Definitions node, then select the Parameters node.

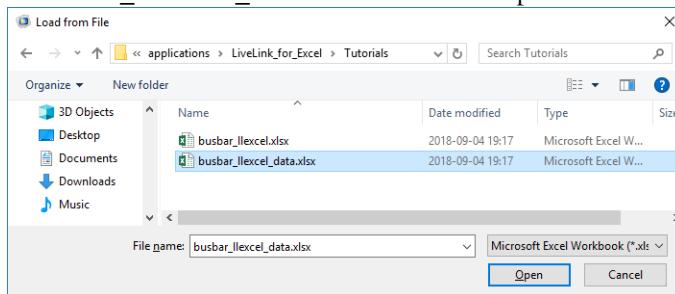
As you can see, the model already contains some parameters in the table.

In order to illustrate the different import capabilities we will this list by a new one.



Name	Expression	Value	Description
L	9[cm]	0.09 m	Length
rad_1	6[mm]	0.006 m	Bolt radius
tbb	5[mm]	0.005 m	Thickness
wbb	5[cm]	0.05 m	Width
mh	3[mm]	0.003 m	Maximum element size
htc	5[W/m^2/K]	5 W/(m <sup>2</sup> ·K)	Heat transfer coefficient
Vtot	20[mV]	0.02 V	Applied voltage

- 2 First clear the table by pressing the Clear Table  button to prevent multiple parameters with the same name.
- 3 To import parameters from an Excel file to the list, click Load from File  located below the table.
- 4 In the Load from File dialog box, select the Microsoft Excel Workbook (\*.xlsx) file type and navigate to the COMSOL installation directory. In the folder applications/LiveLink\_for\_Excel/Tutorials, select the file *busbar\_lllexcel\_data.xlsx* and click Open.



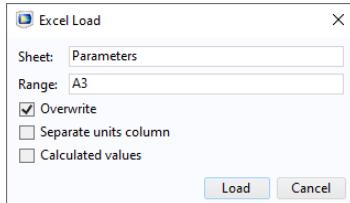
This opens the Excel Load dialog box.

The workbook contains a list of parameters in the Parameters worksheet, which is shown in the figure to the right.

If you compare this list to the parameters defined in the model, you can see that the heat transfer coefficient is not included. The busbar width value is also different; 10 cm in the Excel file instead of 5 cm in the model.

	A	B	C
1	Name	Expression	Description
2	L	9[cm]	Length of busbar
3	rad_1	6[mm]	Radius of bolts
4	tbb	5[mm]	Thickness of busbar
5	wbb	10[cm]	Width of busbar
6	mh	6[mm]	Mesh control
7	Vtot	20[mV]	Voltage

5 In the Excel Load dialog box, specify the sheet and cell range to use for the import. Enter **Parameters** in the Sheet text field, and enter **A3** in the Range text field to import all parameters from the third row. Note that the range should not include the column headers.



6 Click Load.

The parameter list is now imported to the model.

Name	Expression	Value	Description
rad_1	6[mm]	0.006 m	Radius of bolts
tbb	5[mm]	0.005 m	Thickness of busbar
wbb	10[cm]	0.1 m	Width of busbar
mh	6[mm]	0.006 m	Mesh control
Vtot	20[mV]	0.02 V	Voltage

To keep the existing parameters in a model unmodified by the import, you can clear the Overwrite check box in the Excel Load dialog box before the import. In this case, imported parameters will be appended to the table and you will need to resolve any conflicts between similar parameter names yourself.

**Note:** COMSOL does not support multiple parameters with the same name. Only the first parameter definition, from the top, is kept in the table if multiple entries with the same name are present when you leave the Parameters node.

7 In the Home toolbar, click Parameters and select Add>Parameters.

8 In Parameters 2 Settings window, click Load from File .

9 In the Load from File dialog box, select the Microsoft Excel Workbook (\*.xlsx) file type and navigate to the COMSOL installation directory. In the folder `applications/LiveLink_for_Excel/Tutorials`, select the file `busbar_1lexcel_data.xlsx` and click Open.

10 In the Excel Load dialog box, specify the sheet and cell range to use for the import. Enter **Parameters** in the Sheet text field, and enter **A2:C2** in the Range text field to import only the parameter at the second row (L, the length of the busbar).

Here we use a second parameters node only to show how you can link a specify group of parameters with a spreadsheet when running a model from within Excel. See the section [Retrieving and Updating Model Parameters](#).

- 11 In the Geometry toolbar, click Build All  to visualize the geometry with the new parameters value.

## Defining an Interpolation Function Using an Excel® File

---

You will now import data stored in a workbook to define an interpolation function in the model. The function defines the temperature dependency of the heat transfer coefficient between the busbar and the surrounding air.

- 1 On the Home toolbar, click Functions  in the Definitions group. Under the Local section, select Interpolation . On Linux and Mac, the Home toolbar refers to the specific set of controls near the top of the Desktop.
- 2 In the Settings window, under the Definition section, in the Function name text field, enter `htc`.
- 3 Now click Load from File .
- 4 In the Load from File dialog box, select the Microsoft Excel Workbook (\*.xlsx) file type, and navigate to the COMSOL installation directory. In the folder `applications/LiveLink_for_Excel/Tutorials`, select the file `busbar_llexcel_data.xlsx`, then click Open.

The interpolation data is stored in the workbook in a worksheet named `htc(T)`. The temperature values and the corresponding heat transfer coefficient values are defined in two columns, as can be seen in the figure to the right.

	A	B
1	T[K]	htc[W/m^2/K]
2	293	5.0136
3	303	8.5
4	313	9.3272
5	323	9.8918
6	333	10.3303
7	343	10.6921
8	353	11.0012
9	363	11.2717
10	373	11.5123
11	383	11.729
12	393	11.9262

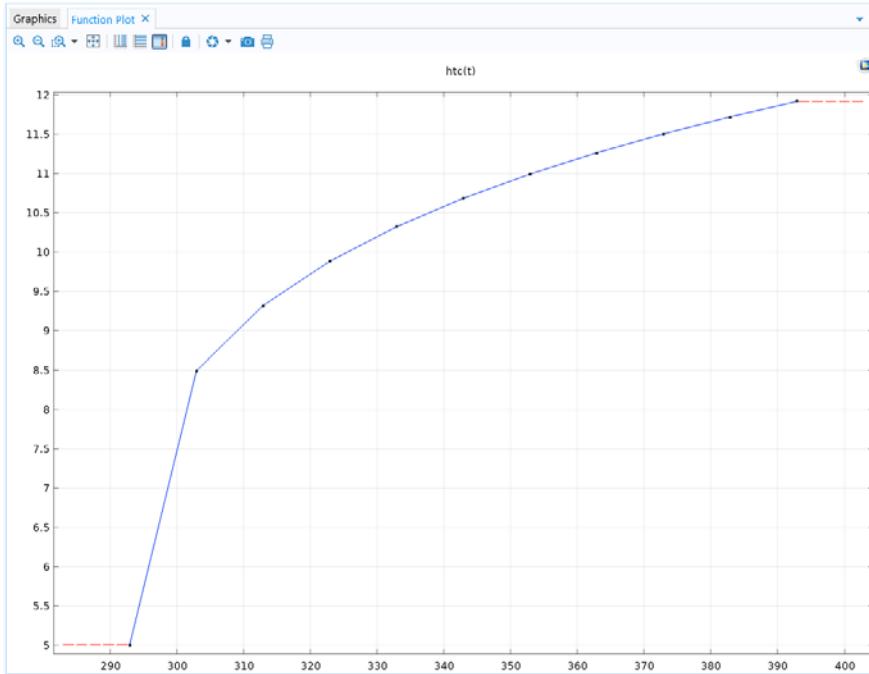
- 5 In the Settings window for Interpolation enter `htc(T)` in the Sheet text field.
- 6 In the Range field enter `A2`.
- 7 To import the data, click Load.

The interpolation table is filled using the data stored in the Excel file.

t	f(t)
293	5.0136
303	8.5
313	9.3272
323	9.8918
333	10.3303
343	10.6921
353	11.0012
363	11.2717
373	11.5123
383	11.729
393	11.9262

- 8 Locate the section, Interpolation and Extrapolation. In the Interpolation list, select Piecewise cubic.
- 9 Locate the section, Units. In the Arguments edit field enter K, in the Function edit field enter  $W / (m^2 * K)$ .

10 To display the interpolation curve, click Plot  .



Now continue with changing the model settings to use the newly defined function for the heat transfer coefficient in the heat flux boundary condition.

- 11 In the Model Builder, under the Component 1 node, expand the Heat Transfer in Solids nodes and then click Heat Flux 1.
- 12 In the Settings window for Heat Flux replace the expression in the Heat transfer coefficient text field with  $htc(T)$ .

## Importing a Sweep List

With the use of an auxiliary sweep, you can solve the model for a range of parameter values, which enables you to vary, for example, a boundary condition. Here, solve the model for different values of the applied voltage  $V_{tot}$ , imported from the same Excel file used previously.

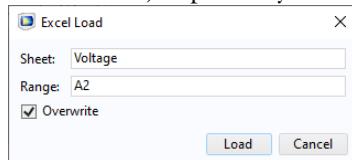
- 1 In the Model Builder, expand Study 1 and click Step 1: Stationary.
- 2 In the Settings window for Stationary expand Study Extensions.

- Under the Study Extensions section, select Auxiliary sweep and then click the Load from File  button.
- In the Load from File dialog box, select the Microsoft Excel Workbook (\*.xlsx) file type, and navigate to the COMSOL Multiphysics installation directory. In the folder `applications/LiveLink_for_Excel/Tutorials`, select the file `busbar_11excel_data.xlsx`, then click Open.

The continuation parameter list can be found in the worksheet, Voltage, in the file.

	A	B	C
1	Continuation parameter	Parameter value list	Unit
2	Vtot	5 10 20 30 40	mV

- In the Excel Load dialog box, enter `Voltage` and `A2`, in the Sheet and Range text fields, respectively.



- Finally, click Load.

The study is now set up to compute the solution for each of the specified voltage values: 5 mV, 10 mV, 20 mV, 30 mV, and 40 mV.

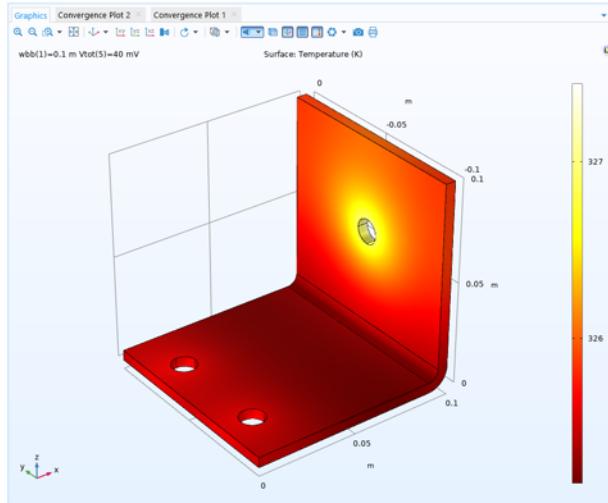
## Computing and Displaying the Solution

---

As a last step, compute the solution that reflects the changes you have applied to the model. But before you will add a maximum operator for postprocessing purpose and a Parametric Sweep node to prepare the model to show how you can link parametric and auxiliary sweep with a spreadsheet when running a model from within Excel. See the section [Running Model in Sweep](#).

- In the Definitions toolbar, click Nonlocal Couplings  and select Maximum .
- In the Maximum 1 settings windows, select Domain 1.
- On the Study toolbar, click Parametric Sweep .
- In the Parametric Sweep node window settings, click Add  and select the parameter wbb (Width of busbar). In the Parameter value list edit field enter `10e-2`.
- On the Study toolbar, click Compute .

- 6 To view the temperature distribution in the busbar only, you need to add a selection to the solution dataset. Under Results > Datasets, right-click Study 1/Parametric Solutions 1 and select Selection.
- 7 In the Settings window for Selection set the Geometric entity level to Domain, and select Domain 1.
- 8 To see the temperature distribution in the busbar for the last parameter value (40 mV), select the Temperature (ht) node. In the 3D plot Group settings window, in the Dataset list, select Study 1/parametric Solutions 1.
- 9 Expand the Temperature (ht) node, in the Surface node settings windows, expands the Range section and clear Manual color range.



- 10 In the Results toolbar click 1D Plot Group .
- 11 In the 1D Plot Group settings window, in Dataset list select Study 1/Parametric Solutions 1.
- 12 In the 1D Plot Group 5 toolbar, click Global  and in the Global settings window, enter `maxop1(T)` in the Expression field.
- 13 Click Plot.

