

INTRODUCTION TO LiveLink™ *for* Excel®

Introduction to LiveLink™ for Excel®

© 2012–2021 COMSOL

Protected by patents listed on 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; 10,776,541, and 11,030,365. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (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.

Version: COMSOL 6.0

Contact Information

Visit the Contact COMSOL page at 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 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. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM023402

Contents

Introduction	5
Working with COMSOL® Models in Excel®	6
Exporting Material Data	27
Loading and Saving Excel® Files from the COMSOL Desktop®	38

Introduction

Using the LiveLink™ *for* Excel®, you can 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, and interpolation data, for example, 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:

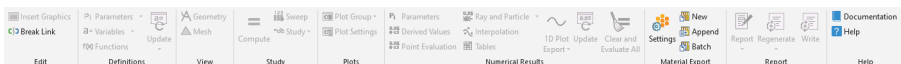
- Opening and saving COMSOL models
- Retrieving and updating Model Parameters
- Computing the solution and extracting results
- Updating existing data in a linked workbook

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](#).

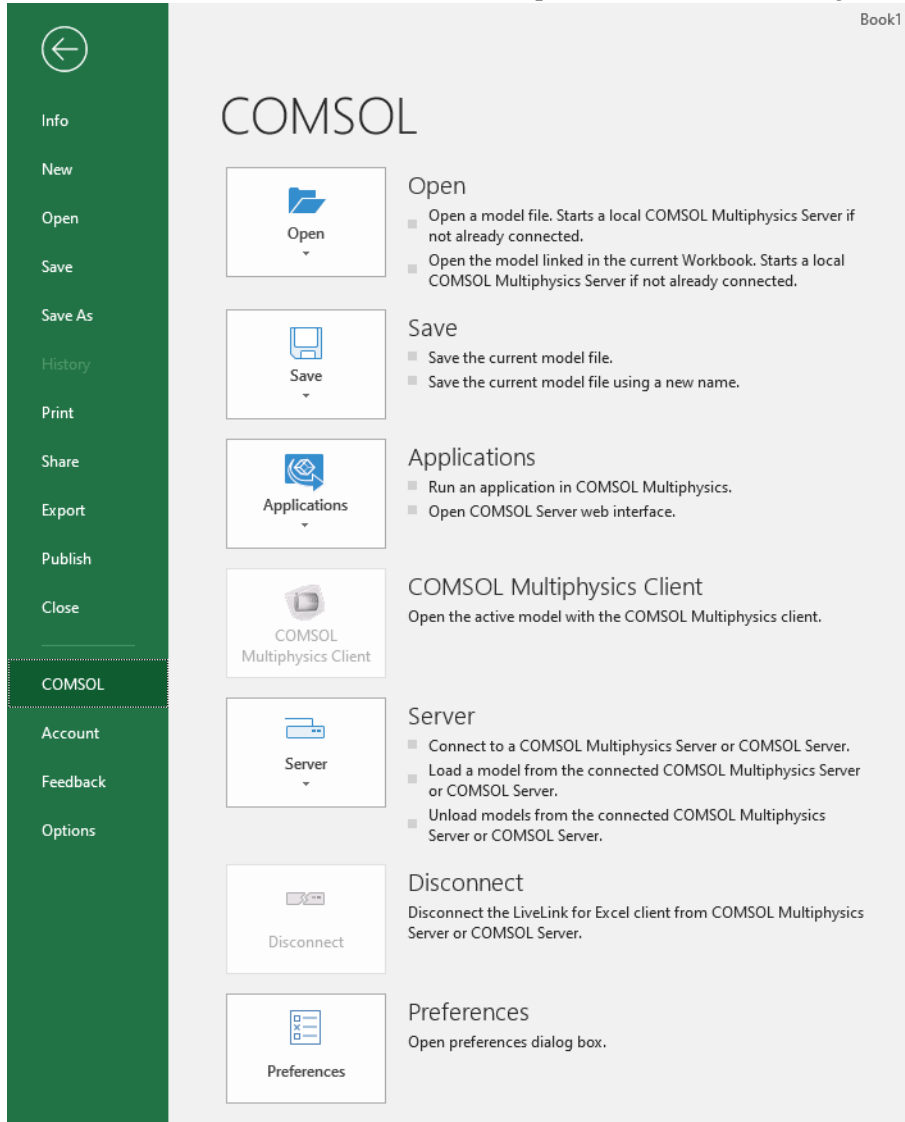
Opening a Model

- 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 6.0 Ribbon tab appears in Excel.



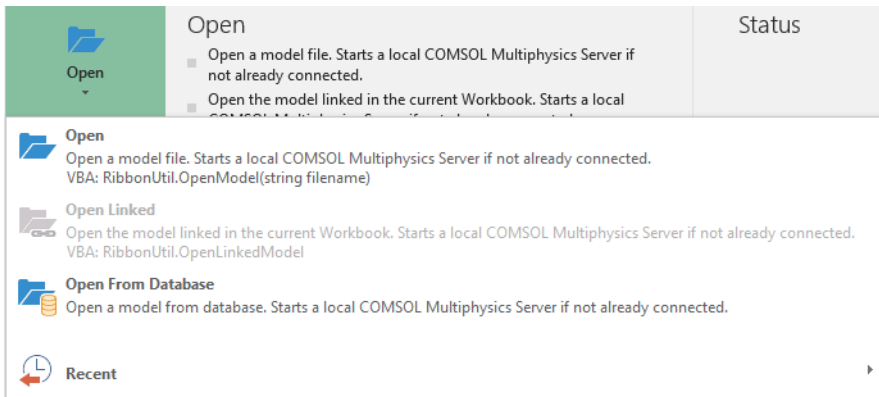
After this you can load a COMSOL model.





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.

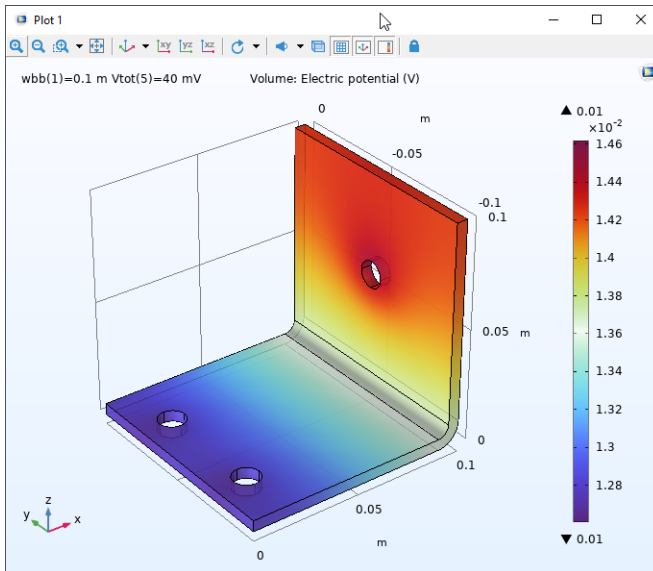
- 4 In the COMSOL backstage view click Open () to expand the open options list.



There you can chose to open a new model file (), open the model linked in the current workbook, if available (), open a model from database (), and open a recent file ().

- 5 As we start from blank worksheet and want to lod a model from the COMSOL Application library, click Open ().
- 6 In your COMSOL installation directory, find the folder applications/ LiveLink_for_Excel/Tutorials.
- 7 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_1\excel.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 6.0 tab, Definitions group, click Parameters **P_i**.

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	I
1	COMSOL Model								
2	Filename:	Multiphysics\applications\LiveLink_for_Excel\Tutorials\busbar_IIexcel.mph							
3									
4	Parameter	Expression	Unit	Descriptive	Value	Unit			
5	rad_1	6 mm	Radius of	0.006	m				
6	tbb	5 mm	Thickness	0.005	m				
7	wbb	10 cm	Width of b	0.1	m				
8	mh	6 mm	Mesh con	0.006	m				
9	Vtot	2.00E+01 mV	Voltage	0.02	V				
10	L	9 cm	Length of	0.09	m				
11									

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 6.0 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	I
1	COMSOL Model								
2	Filename:	Multiphysics\applications\LiveLink_for_Excel\Tutorials\busbar_IIexcel.mph							
3									
4	Parameter	Expression	Unit	Descriptive	Value	Unit			
5	Parameters 2								
6	L	9 cm	Length	0.09	m				
7									

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:

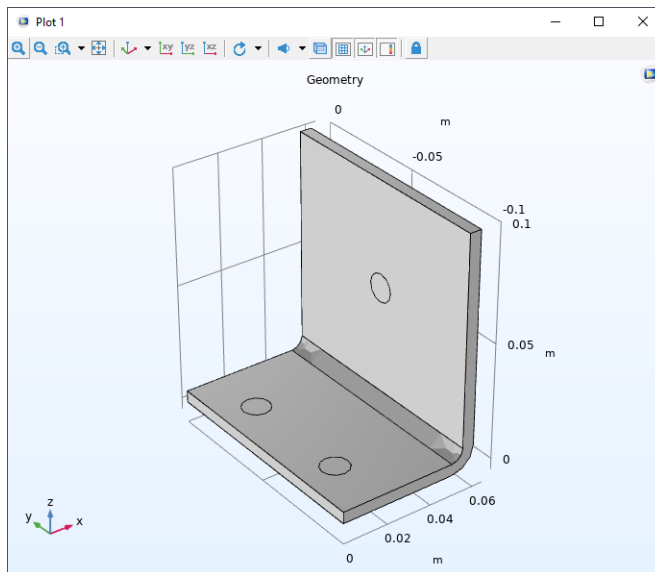
- 5 In cell B6 enter 5, the parameter L is then automatically updated to 5 in COMSOL Multiphysics.

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:




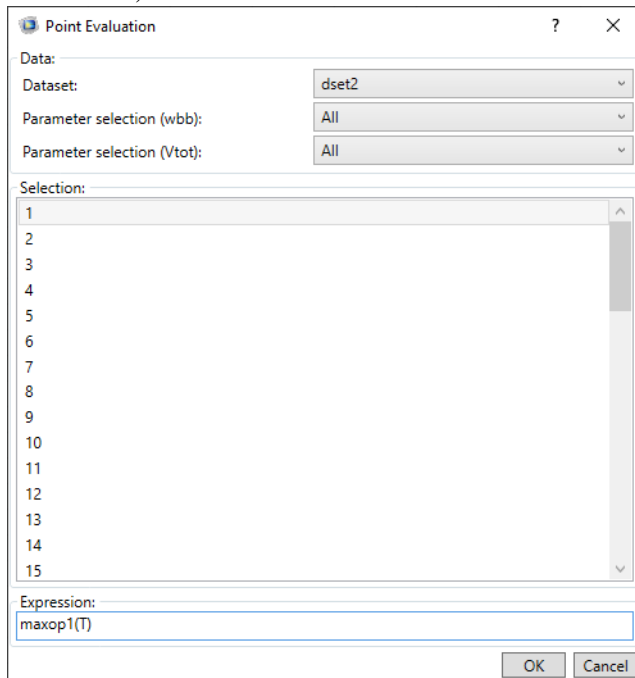
- 7 You will now set the value of parameter L back to 9 cm, in cell B6 enter 9 and press Enter.

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 A13.
- 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
1	COMSOL Model								
2	Filename: Multiphysics\applications\LiveLink_for_Excel\Tutorials\busbar_llexcel.mph								
3									
4	Parameter Expression	Unit	Descriptive	Value	Unit				
5	Parameters 2								
6	L	9 cm	Length	0.09 m					
7									
8									
9									
10									
11									
12									
13	wbb (m)	Vtot (mV) Maximum (K), Point: 1							
14	0.1	5	294.0778						
15	0.1	10	296.3628						
16	0.1	20	303.1653						
17	0.1	30	313.7691						
18	0.1	40	327.4482						
19									

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.

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

- 3 Select cell D3, then go to the Results group and click Interpolation .

Interpolation ? X

Dataset:
Study 1/Parametric Solutions 1 (dset2)

Parameter:
wbb = 0.1

Expression:
ht.Qtot


Coordinates from cell range:
A3:C20

☐ Include Header

OK Cancel

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					
3	0	0	0	0.000286	0.001144	0.004574	0.010292	0.018297
4	2.50E-02	0	0	82.18433	328.7373	1314.949	2958.636	5259.797
5	5.00E-02	0	0	117.5953	470.3813	1881.525	4233.432	7526.101
6	0	-1.25E-02	0	18.24033	72.96132	291.8453	656.6518	1167.381
7	2.50E-02	-1.25E-02	0	118.3052	473.2207	1892.883	4258.986	7571.531
8	5.00E-02	-1.25E-02	0	119.7238	478.8954	1915.582	4310.059	7662.326
9	0	-2.50E-02	0	0.010623	0.04249	0.169961	0.382411	0.679843
10	2.50E-02	-2.50E-02	0	201.1331	804.5326	3218.13	7240.793	12872.52
11	5.00E-02	-2.50E-02	0	122.3667	489.4667	1957.867	4405.2	7831.467
12	0	0	5.00E-03	0.000238	0.000953	0.003813	0.00858	0.015254
13	2.50E-02	0	5.00E-03	81.59123	326.3649	1305.46	2937.284	5221.839
14	5.00E-02	0	5.00E-03	117.5883	470.3531	1881.412	4233.178	7525.649
15	0	-1.25E-02	5.00E-03	17.63211	70.52844	282.1138	634.756	1128.455
16	2.50E-02	-1.25E-02	5.00E-03	118.3924	473.5697	1894.279	4262.127	7577.115
17	5.00E-02	-1.25E-02	5.00E-03	119.7715	479.0861	1916.344	4311.775	7665.378
18	0	-2.50E-02	5.00E-03	0.047304	0.189215	0.756858	1.702932	3.027434
19	2.50E-02	-2.50E-02	5.00E-03	154.7453	618.981	2475.924	5570.829	9903.696
20	5.00E-02	-2.50E-02	5.00E-03	122.3256	489.3025	1957.21	4403.722	7828.839
21								


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.

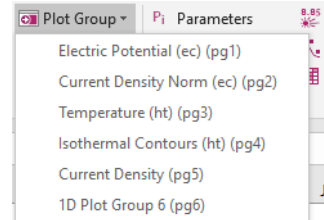
- 8 You can format the worksheet for instance include column labels, insert cell at your convenience. Make sure that the cell comment remain in the worksheet to allow future update.

	A	B	C	D	E	F	G	H
1				Qtot[W]				
2	x	y	z	5mV	10mV	20mV	30mV	40mV
3	0	0	0	0.000286	0.001144	0.004574	0.010292	0.018297
4	2.50E-02	0	0	82.18433	328.7373	1314.949	2958.636	5259.797
5	5.00E-02	0	0	117.5953	470.3813	1881.525	4233.432	7526.101
6	0	-1.25E-02	0	18.24033	72.96132	291.8453	656.6518	1167.381
7	2.50E-02	-1.25E-02	0	118.3052	473.2207	1892.883	4258.986	7571.531
8	5.00E-02	-1.25E-02	0	119.7238	478.8954	1915.582	4310.059	7662.326
9	0	-2.50E-02	0	0.010623	0.04249	0.169961	0.382411	0.679843
10	2.50E-02	-2.50E-02	0	201.1331	804.5326	3218.13	7240.793	12872.52
11	5.00E-02	-2.50E-02	0	122.3667	489.4667	1957.867	4405.2	7831.467
12	0	0	5.00E-03	0.000238	0.000953	0.003813	0.00858	0.015254
13	2.50E-02	0	5.00E-03	81.59123	326.3649	1305.46	2937.284	5221.839
14	5.00E-02	0	5.00E-03	117.5883	470.3531	1881.412	4233.178	7525.649
15	0	-1.25E-02	5.00E-03	17.63211	70.52844	282.1138	634.756	1128.455
16	2.50E-02	-1.25E-02	5.00E-03	118.3924	473.5697	1894.279	4262.127	7577.115
17	5.00E-02	-1.25E-02	5.00E-03	119.7715	479.0861	1916.344	4311.775	7665.378
18	0	-2.50E-02	5.00E-03	0.047304	0.189215	0.756858	1.702932	3.027434
19	2.50E-02	-2.50E-02	5.00E-03	154.7453	618.981	2475.924	5570.829	9903.696
20	5.00E-02	-2.50E-02	5.00E-03	122.3256	489.3025	1957.21	4403.722	7828.839

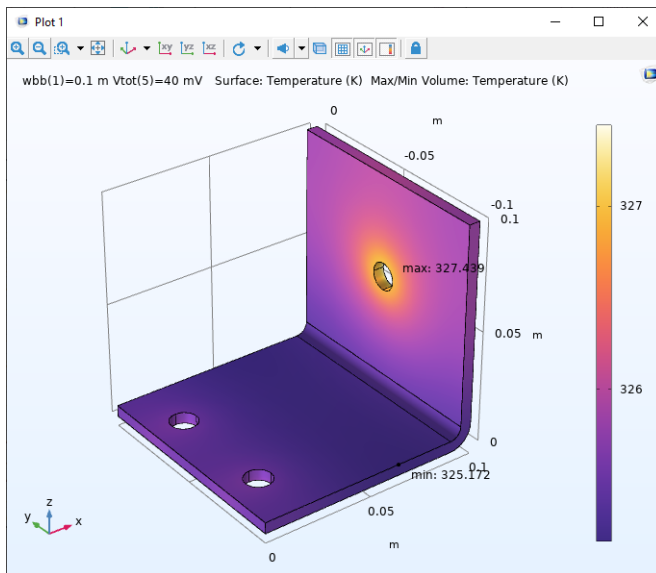
Displaying the Solution

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.


- 1 Click the Plot Group button , from the Plots group, to select the plot group to display. Here, select Temperature (ht) (pg3).




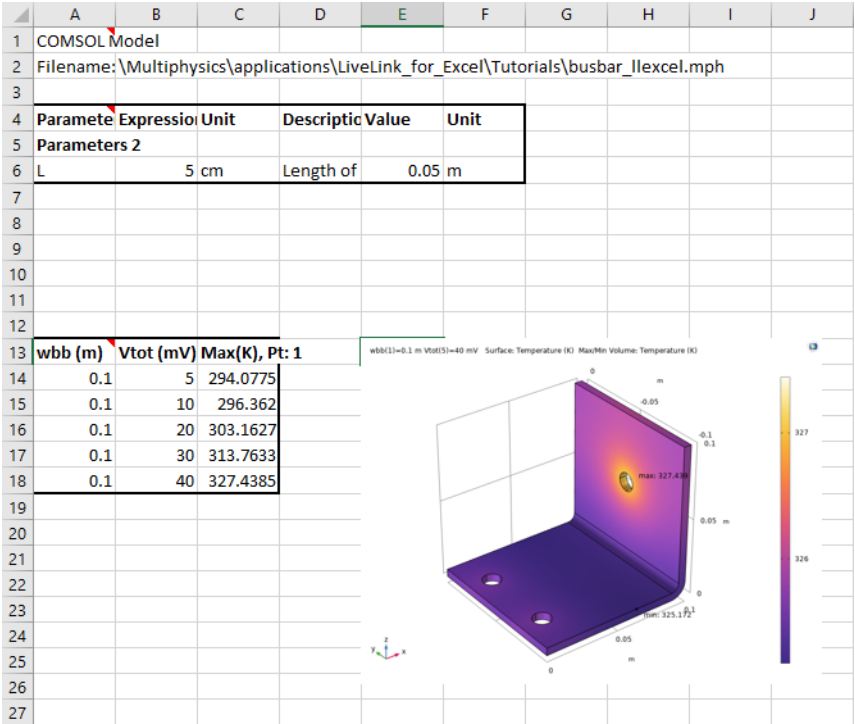
The graphics window now displays the temperature distribution in the busbar as in the figure below:



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.

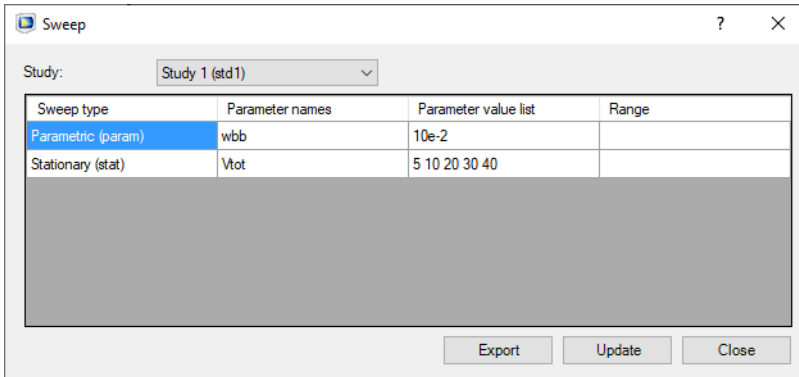
To insert the displayed image into the worksheet, in Sheet 1 select cell E13, then click Insert Graphics  from the Edit group.




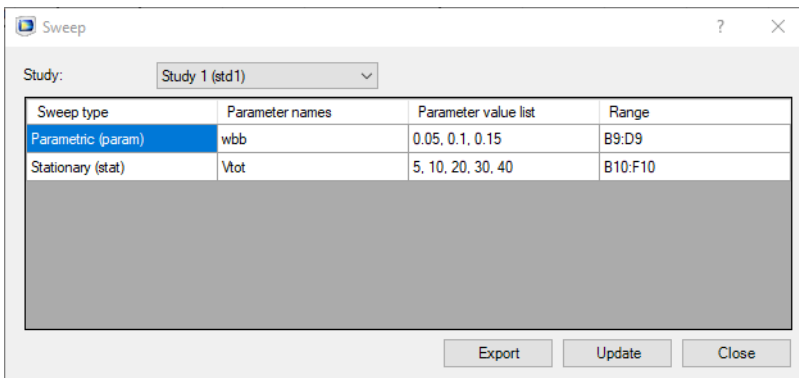
Running Model in Sweep

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

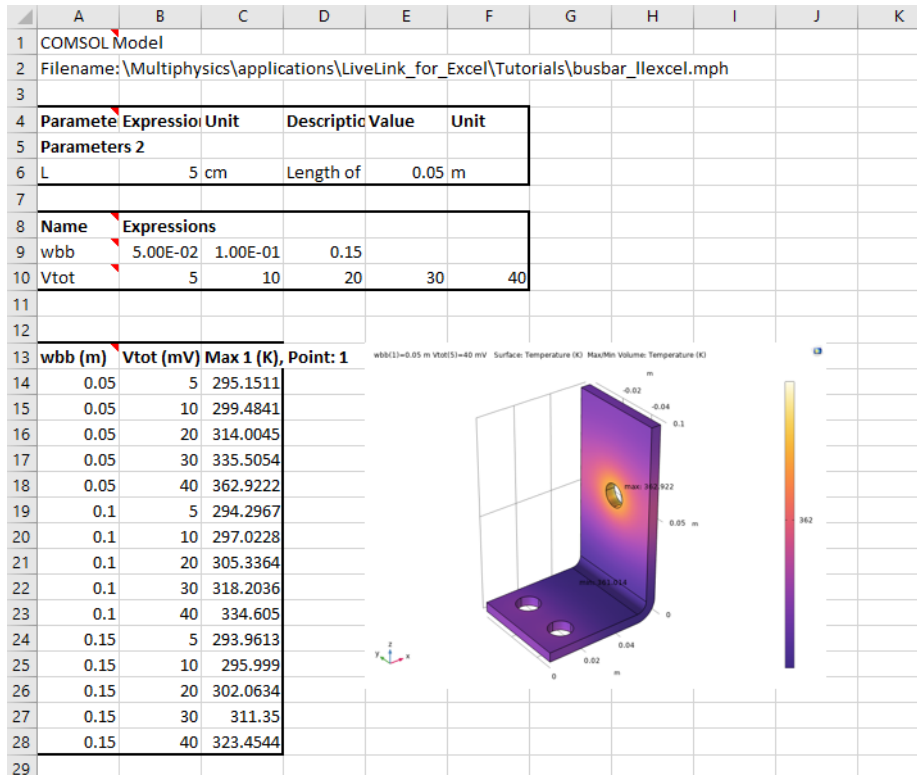
Now that you have a workbook containing model definitions and data evaluation linked to a COMSOL model, you can easily modify and recompute the model from the workbook, while keeping the evaluated results up to date.

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.




- 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.

- In the Numerical Results group click Update .

- 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	5mV	10mV	20mV	30mV	40mV
3	0	0	0	0.179461	0.717846	2.871383	6.460611	11.48553
4	2.50E-02	0	0	501.054	2004.216	8016.864	18037.94	32067.46
5	5.00E-02	0	0	318.2405	1272.962	5091.848	11456.66	20367.39
6	0	-1.25E-02	0	0.004354	0.017415	0.069659	0.156733	0.278637
7	2.50E-02	-1.25E-02	0	444.3071	1777.228	7108.914	15995.06	28435.66
8	5.00E-02	-1.25E-02	0	314.612	1258.448	5033.792	11326.03	20135.17
9	0	-2.50E-02	0	0.002706	0.010825	0.043299	0.097423	0.173196
10	2.50E-02	-2.50E-02	0	501.8369	2007.347	8029.39	18066.13	32117.56
11	5.00E-02	-2.50E-02	0	315.2152	1260.861	5043.444	11347.75	20173.77
12	0	0	5.00E-03	0.02029	0.081162	0.324647	0.730455	1.298587
13	2.50E-02	0	5.00E-03	498.7006	1994.803	7979.21	17953.22	31916.84
14	5.00E-02	0	5.00E-03	730.4337	2921.735	11686.94	26295.61	46747.76
15	0	-1.25E-02	5.00E-03	0.029851	0.119406	0.477622	1.07465	1.910489
16	2.50E-02	-1.25E-02	5.00E-03	375.3431	1501.373	6005.49	13512.35	24021.96
17	5.00E-02	-1.25E-02	5.00E-03	740.7044	2962.818	11851.27	26665.36	47405.08
18	0	-2.50E-02	5.00E-03	0.026326	0.105303	0.421211	0.947725	1.684845
19	2.50E-02	-2.50E-02	5.00E-03	499.1806	1996.723	7986.89	17970.5	31947.56
20	5.00E-02	-2.50E-02	5.00E-03	728.1643	2912.657	11650.63	26213.91	46602.51

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.

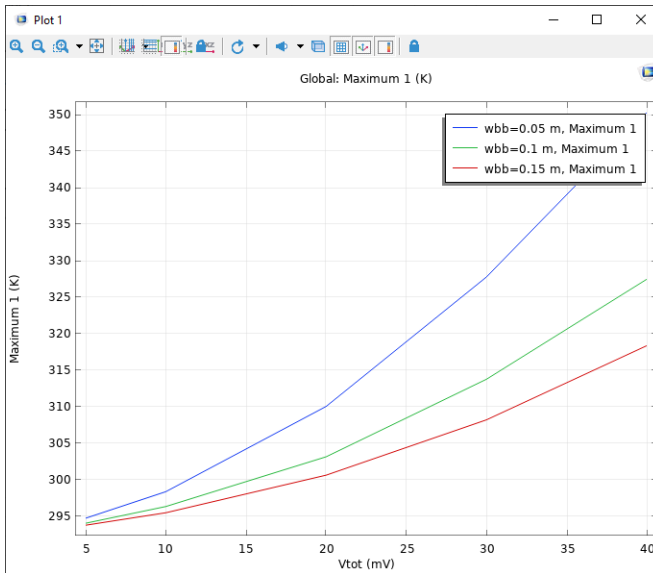
	A	B	C	D	E	F	G	H	I	J	K	L	M
1				Q _{tot} [W] (wbb=0.05m)					Q _{tot} [W] (wbb=0.1m)				
2	x	y	z	5mV	10mV	20mV	30mV	40mV	5mV	10mV	20mV	30mV	40mV
3	0	0	0	0.179461	0.717846	2.871383	6.460611	11.48553	0.000395	0.001579	0.006314	0.014207	0.025257
4	2.50E-02	0	0	501.054	2004.216	8016.864	18037.94	32067.46	81.92444	327.6978	1310.791	2949.28	5243.164
5	5.00E-02	0	0	318.2405	1272.962	5091.848	11456.66	20367.39	80.06234	320.2494	1280.997	2882.244	5123.99
6	0	-1.25E-02	0	0.004354	0.017415	0.069659	0.156733	0.278637	18.37656	73.50623	294.0249	661.556	1176.1
7	2.50E-02	-1.25E-02	0	444.3071	1777.228	7108.914	15995.06	28435.66	117.1475	468.59	1874.36	4217.31	7497.44
8	5.00E-02	-1.25E-02	0	314.612	1258.448	5033.792	11326.03	20135.17	78.51356	314.0542	1256.217	2826.488	5024.868
9	0	-2.50E-02	0	0.002706	0.010825	0.043299	0.097423	0.173196	0.034535	0.138141	0.552565	1.243271	2.210259
10	2.50E-02	-2.50E-02	0	501.8369	2007.347	8029.39	18066.13	32117.56	212.1263	848.5054	3394.022	7636.549	13576.09
11	5.00E-02	-2.50E-02	0	315.2152	1260.861	5043.444	11347.75	20173.77	83.22106	332.8842	1331.537	2995.958	5326.148
12	0	0	5.00E-03	0.02029	0.081162	0.324647	0.730455	1.298587	6.19E-05	0.000248	0.000991	0.00223	0.003964
13	2.50E-02	0	5.00E-03	498.7006	1994.803	7979.21	17953.22	31916.84	81.81896	327.2758	1309.103	2945.483	5236.413
14	5.00E-02	0	5.00E-03	730.4337	2921.735	11686.94	26295.61	46747.76	184.1176	736.4702	2945.881	6628.232	11783.52
15	0	-1.25E-02	5.00E-03	0.029851	0.119406	0.477622	1.07465	1.910489	17.9838	71.9352	287.7408	647.4168	1150.963
16	2.50E-02	-1.25E-02	5.00E-03	375.3431	1501.373	6005.49	13512.35	24021.96	118.2093	472.8373	1891.349	4255.536	7565.397
17	5.00E-02	-1.25E-02	5.00E-03	740.7044	2962.818	11851.27	26665.36	47405.08	173.9964	695.9856	2783.942	6263.871	11135.77
18	0	-2.50E-02	5.00E-03	0.026326	0.105303	0.421211	0.947725	1.684845	0.006999	0.027997	0.111989	0.251975	0.447956
19	2.50E-02	-2.50E-02	5.00E-03	499.1806	1996.723	7986.89	17970.5	31947.56	157.154	628.6161	2514.464	5657.545	10057.86
20	5.00E-02	-2.50E-02	5.00E-03	728.1643	2912.657	11650.63	26213.91	46602.51	189.0866	756.3466	3025.386	6807.119	12101.55

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


Importing 1D Plot Data

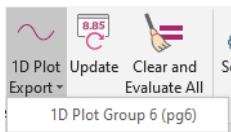
Excel does not support general 2D- and 3D-plots that are commonly produced in COMSOL. However, 1D-plots have data that can be extracted to Excel and Excel can create plots of these data as well.

In the busbar_llexcel model there is a single 1D-plot that looks like this when plotted using the COMSOL graphics server:

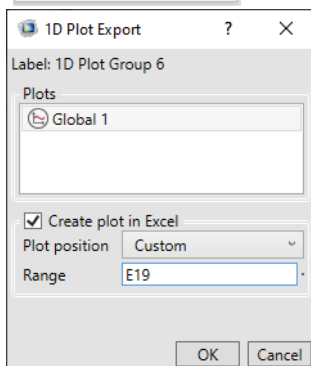


This plot can be inserted into Excel as a static image. Instead we will insert the data that makes up the plot and create a scatter chart in the Excel sheet.

- 1 In Sheet 1 select cell E13, 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).



- 2 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 6 (pg6).

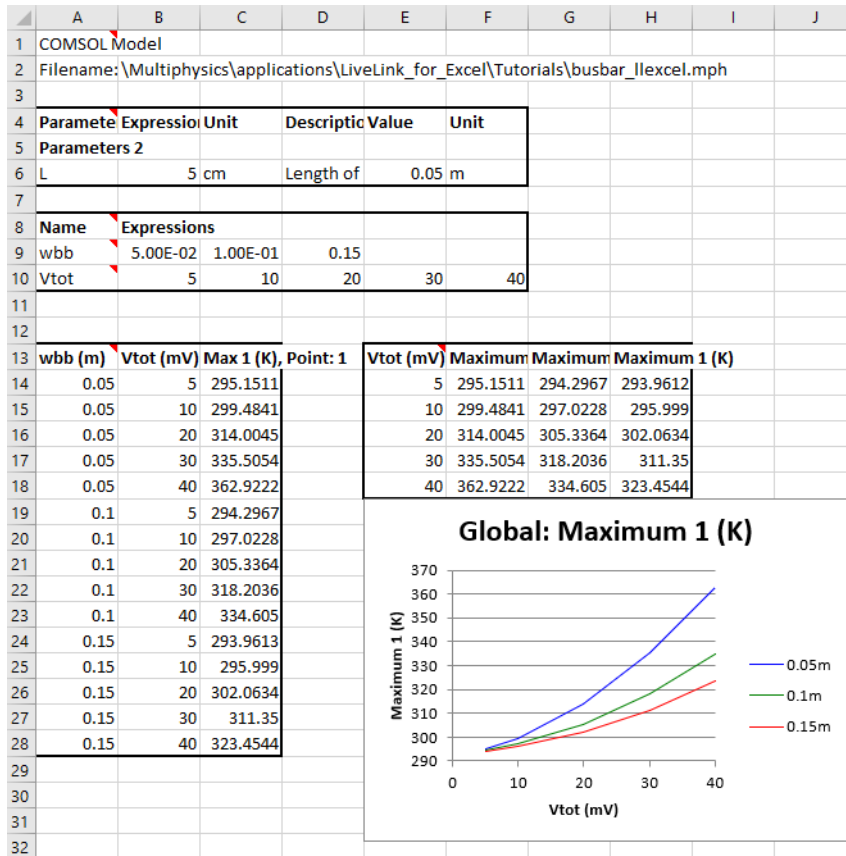


- 3 This opens a dialog box when you can set the data to be imported into Excel.

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.



4 In Plot position list, select Custom and enter E19 in the Range edit field.

The final chart looks like this when using the default settings:

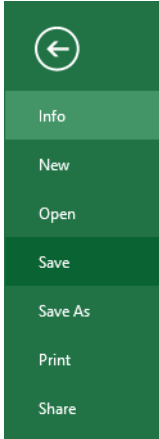


Saving the Model and the Workbook

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.

- 1 Go to the Excel File menu and select COMSOL. In the COMSOL backstage view click Save button  and then 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_llexcel.xlsm` 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.

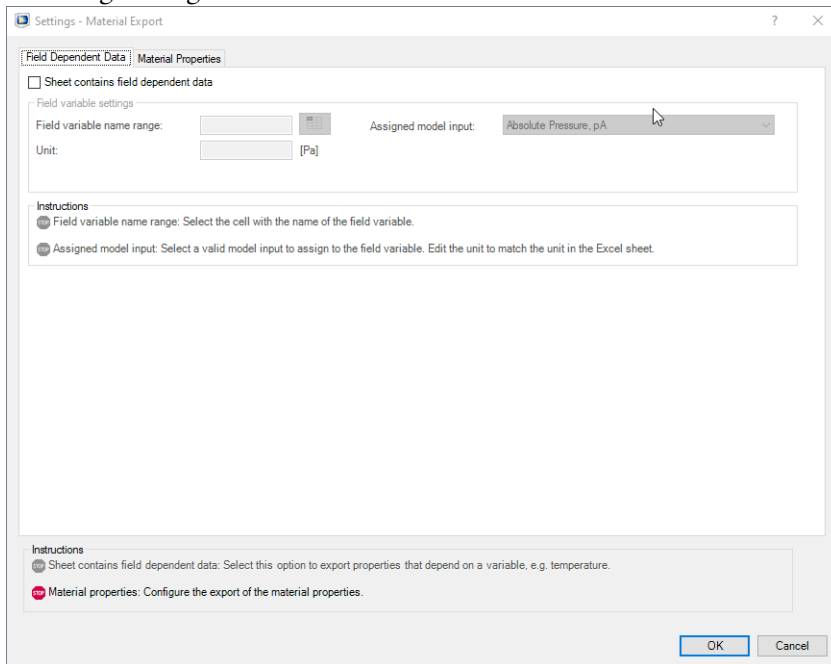
	A	B	C
1		thermal conductivity (W/m/K)	density (lb/in^3)
2	Copper	400	3.14E-01
3	Titanium	10	1.79E-01
4	Aluminum	150	9.75E-02

- 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/in³.

	A	B	C	D	E	F
1		thermal conductivity (W/m/K)	density (lb/in^3)	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

- On the COMSOL 6.0 tab, click Settings  to open the Material Export Settings dialog box.




Settings - Material Export

Field Dependent Data | Material Properties


☐ Sheet contains field dependent data


Field variable settings

Field variable name range:  Assigned model input: Absolute Pressure, pA


Unit: [Pa]


Instructions

 Field variable name range: Select the cell with the name of the field variable.

 Assigned model input: Select a valid model input to assign to the field variable. Edit the unit to match the unit in the Excel sheet.

Instructions

 Sheet contains field dependent data: Select this option to export properties that depend on a variable, e.g. temperature.

 Material properties: Configure the export of the material properties.

OK Cancel

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.

The screenshot shows the 'Settings - Material Export' dialog box with the 'Material Properties' tab selected. The 'Field Dependent Data' tab is also visible. The 'Data range' section has two text fields: 'Material names range:' and 'Property names range:', each with a 'Range' button to its right. The 'Material property settings' section contains a large empty table. To the right of this table is an 'Assign' button and a 'Valid properties (SI Units)' list. The list has a 'Filter:' text field and a scrollable list of property categories: Acoustics, Basic Properties, Electrochemistry, Electromagnetic Models, Gas models, Piezoelectric models, Piezoresistive models, Semiconductors, and Solid mechanics. Below the list is a 'Unit:' text field. At the bottom, there are two instruction sections. The first section has a 'Sheet contains field dependent data' checkbox and two instructions. The second section has a 'Material properties' checkbox and one instruction. 'OK' and 'Cancel' buttons are at the bottom right.

Settings - Material Export

Field Dependent Data | Material Properties

Data range

Material names range:

Property names range:

Material property settings

Valid properties (SI Units)

Filter:

- [-] Acoustics
- [-] Basic Properties
- [-] Electrochemistry
- [-] Electromagnetic Models
- [-] Gas models
- [-] Piezoelectric models
- [-] Piezoresistive models
- [-] Semiconductors
- [-] Solid mechanics

Unit:

Assign

Instructions

[-] Sheet contains field dependent data: Select this option to export properties that depend on a variable, e.g. temperature.

[-] Material properties: Configure the export of the material properties.

OK Cancel

- 5 You will now select the cells containing the material names. Click the Range button () 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 () to the right of the Property names range text field.
- 8 Select the range B1:F1 and click OK.

This screenshot shows the same 'Settings - Material Export' dialog box, but now the 'Material names range' field contains 'A2:A5' and the 'Property names range' field contains 'B1:F1'. The 'Range' buttons are still present. The 'Valid properties (SI Units)' list is still visible on the right. The 'OK' and 'Cancel' buttons are at the bottom right.

Settings - Material Export

Field Dependent Data | Material Properties

Data range

Material names range: Copper, Titanium, Aluminum, Steel

Property names range:

Material property settings

Valid properties (SI Units)

Filter:

- [-] Acoustics
- [-] Basic Properties
- [-] Electrochemistry
- [-] Electromagnetic Models
- [-] Gas models
- [-] Piezoelectric models
- [-] Piezoresistive models
- [-] Semiconductors
- [-] Solid mechanics

Unit:

Assign

Instructions

[-] Sheet contains field dependent data: Select this option to export properties that depend on a variable, e.g. temperature.

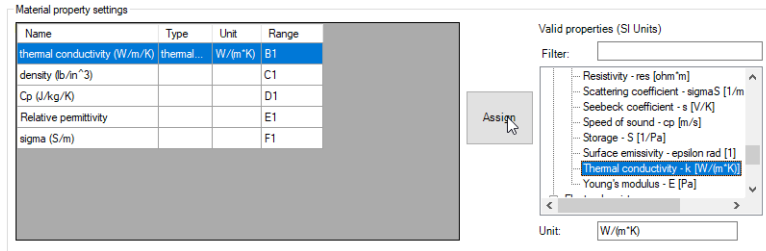
[-] Material properties: Configure the export of the material properties.

OK Cancel

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.



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

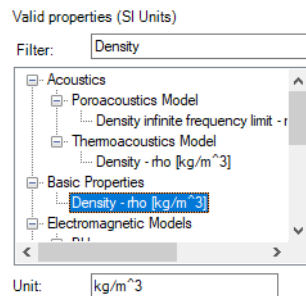
11 Now in the Filter edit field enter Density to search for all available properties density type.

12 select density (lb/in^3) from the Name column, then select Density - rho [kg/m^3] 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.

13 Below the valid properties list, locate the Unit text field and enter lb/in^3.

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



15 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.

Settings - Material Export

Field Dependent Data | Material Properties

Data range
Material names range: A2:A5
Property names range: B1:F1

Material property settings

Name	Type	Unit	Range
thermal conductivity (W/m/K)	thermal...	W/m ² K	B1
density (lb/in ³)	density	lb/in ³	C1
Cp (J/kg/K)	heatcap...	J/(kg*K)	D1
Relative permittivity	relperm...	1	E1
sigma (S/m)	electric...	S/m	F1

Valid properties (SI Units)

Filter:

- Diffusion coefficient - D [m²/s]
- Dynamic viscosity - mu [Pa*s]
- Electrical conductivity - sigma [S/m]
- Electron mobility - mue [m²/(V*s)]
- Extinction coefficient - betaR [1/m]
- Frequency factor - alpha [1/s]
- Heat capacity at constant pressure - Cp
- Isotropic structural loss factor - eta s [1]

Assign

Unit: S/m

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.

Instructions

- Sheet contains field dependent data: Select this option to export properties that depend on a variable, e.g. temperature.
- ✓ Material properties: Configure the export of the material properties.

OK Cancel

16 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^3)	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

1 In the workbook `busbar_11excel_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 6.0 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.

The screenshot shows the 'Settings - Material Export' dialog box with the 'Material Properties' tab selected. The 'Field Dependent Data' section is checked. Under 'Field variable settings', the 'Field variable name range' is set to 'A2' and the 'Unit' is set to 'K'. The 'Assigned model input' is set to 'Temperature, T'. The 'Instructions' section contains two green checkmarks: 'Field variable name range: Select the cell with the name of the field variable.' and 'Assigned model input: Select a valid model input to assign to the field variable. Edit the unit to match the unit in the Excel sheet.'

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 (📊) 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 (📊) 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"/>

Assign

Valid properties (SI Units)

Filter:

- 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 - Mf [kg/(m^2*s)]

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.

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

13 Repeat steps 11 and 12 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)
rho(kg/m^3)	Density - rho [kg/m^3]	kg/m^3
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.

Settings - Material Export

Field Dependent Data | Material Properties

Data range
Material names range: A1 Water, liquid
Property names range: B2:E2

Material property settings

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

Assign

Valid properties (SI Units)
Filter: thermal co

- Acoustics
 - Thermoacoustics Model
 - Thermal conductivity - k [W/(m*K)]
- Basic Properties
 - Thermal conductivity - k [W/(m*K)]

Unit: W/(m*K)

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.

Instructions

- ✓ Sheet contains field dependent data: Select this option to export properties that depend on a variable, e.g. temperature.
- ✓ Material properties: Configure the export of the material properties.

OK Cancel

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.



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

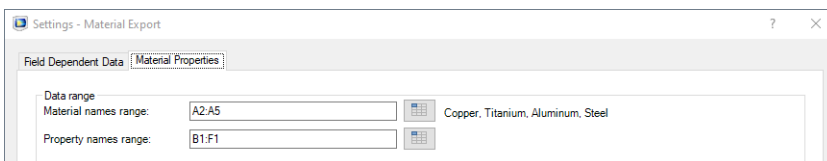
- 15 Note that comments now appear on the cells containing the material names and properties.

	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/v60/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/v60/materials folder.



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

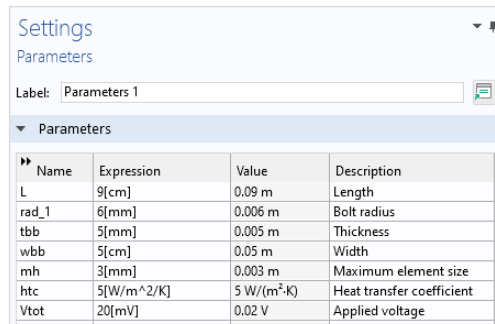
- 1 If it is not already open, start a new COMSOL Desktop. From the File toolbar, select Application Libraries .
- 2 In the Application Libraries window, choose COMSOL Multiphysics>Multiphysics>busbar and click Open .

Importing Parameters from Excel®

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

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



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

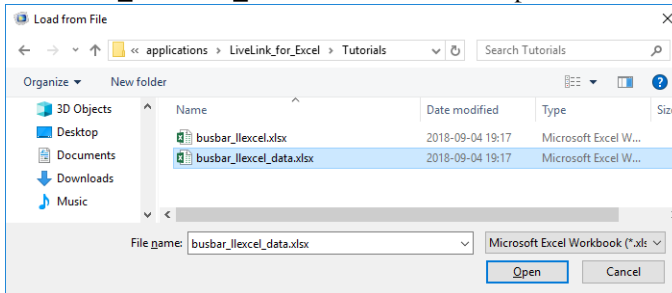


Settings
Parameters

Label: Parameters 1

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 ² ·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_llexcel_data.xlsx and click Open.



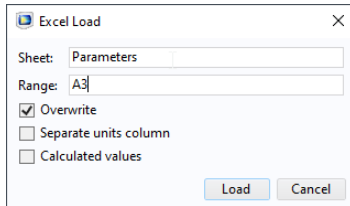
This opens the Excel Load dialog box.

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

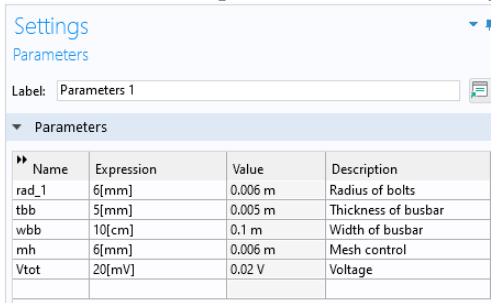
If you compare this list to the parameters defined in the model, you can see that the heat transfer coefficient is not included. The busbar width value is also different; 10 cm in the Excel file instead of 5 cm in the model.

	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.




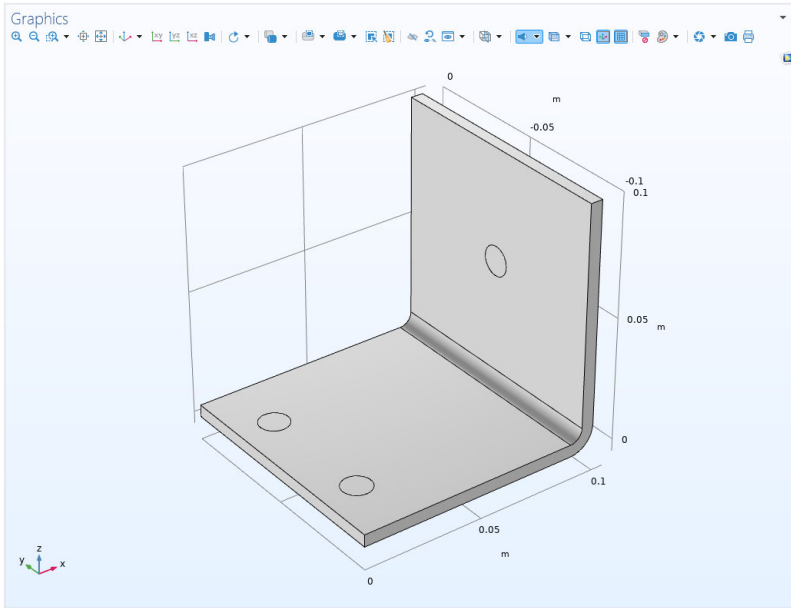
To keep the existing parameters in a model unmodified by the import, you can clear the Overwrite check box in the Excel Load dialog box before the import. In this case, imported parameters will be appended to the table and you will need to resolve any conflicts between similar parameter names yourself.

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](#).

- II In the Geometry toolbar, click Build All  and go to Geometry 1 node 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_1lexcel_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.

Settings
Interpolation
Plot Create Plot

Label: Interpolation 1

Definition

Data source: Local table

Function name: htc

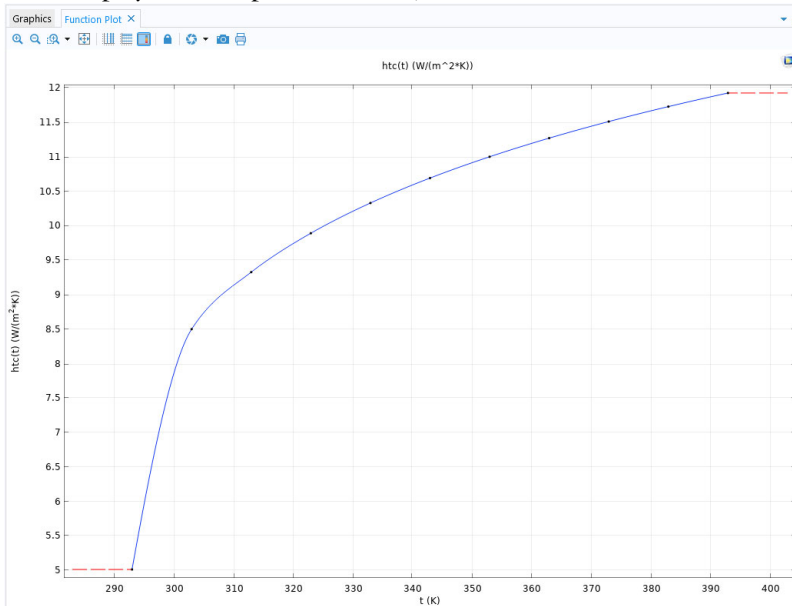
t	f(t)
293	5.0136
303	8.5
313	9.3272
323	9.8918
333	10.3303
343	10.6921
353	11.0012
363	11.2717
373	11.5123
383	11.729
393	11.9262

- 8 Locate the section, Interpolation and Extrapolation. In the Interpolation list, select Piecewise cubic.

- 9 Locate the section, Units. In the Function text field enter $W/(m^2 \cdot K)$. in the Arguments text field enter K.

Units	
Function	Unit
htc	$W/(m^2 \cdot K)$
Argument	Unit
t	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

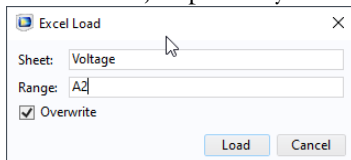
With the use of an auxiliary sweep, you can solve the model for a range of parameter values, which enables you to vary, for example, a boundary condition. Here, solve the model for different values of the applied voltage V_{tot} , imported from the same Excel file used previously.

- 1 In the Model Builder, expand Study 1 and click Step 1: Stationary.
- 2 In the Settings window for Stationary expand Study Extensions.
- 3 Under the Study Extensions section, select Auxiliary sweep and then click the Load from File  button.
- 4 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_1lexcel_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

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







- 6 Finally, click Load.

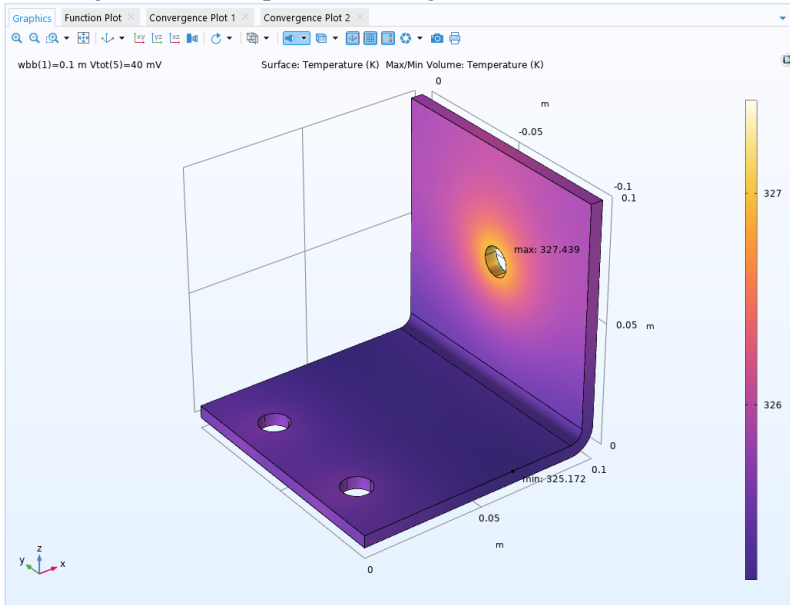
The study is now set up to compute the solution for each of the specified voltage values: 5 mV, 10 mV, 20 mV, 30 mV, and 40 mV.




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](#).

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

- 5 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, and select Surface. 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 6 toolbar, click Global  and in the Global settings window, enter $\max_{op1}(T)$ in the Expression field.
- 13 Click Plot .

