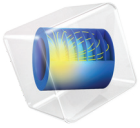


INTRODUCTION TO LiveLink™ *for* Excel®

5.5

COMSOL

Introduction to LiveLink™ *for* Excel®

Version: COMSOL 5.5

Contact Information

Visit the Contact COMSOL page at 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 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/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM023402

Contents

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:

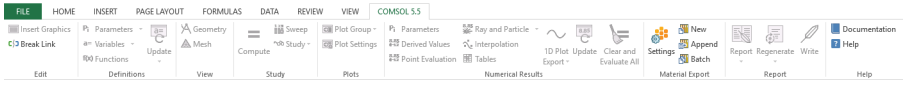
- Opening and saving COMSOL models
- Retrieving and updating Model Parameters
- Computing the solution and extracting results
- Updating existing data in a linked workbook

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.

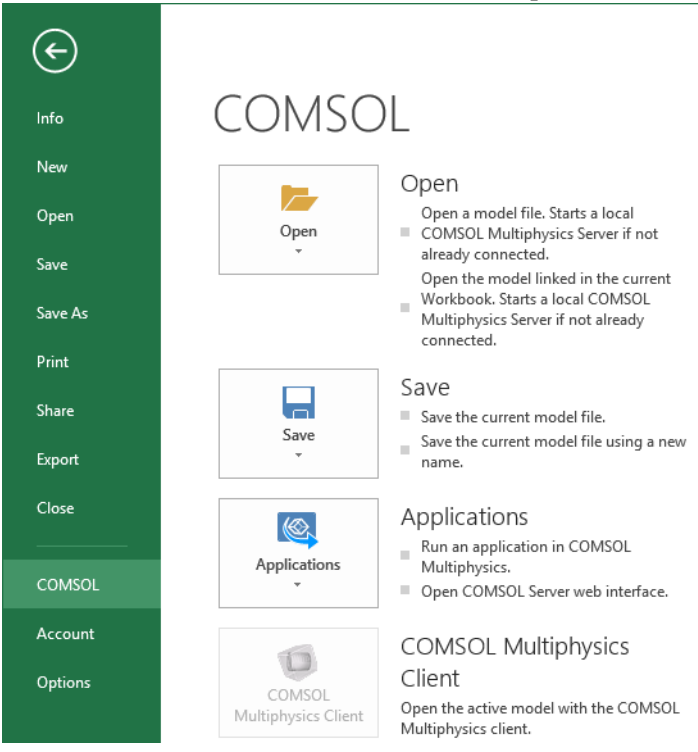
Opening a Model

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 it that the COMSOL 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.



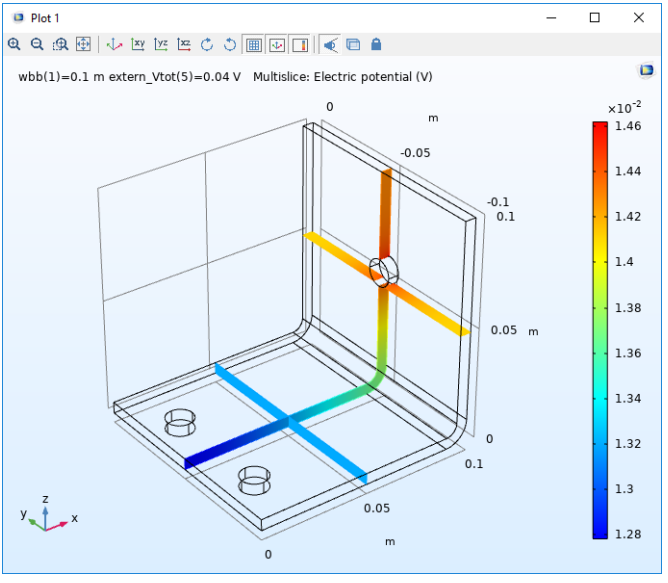
Note: In the COMSOL backstage view you can manage the COMSOL Multiphysics model, launch COMSOL Apps, connect manually to a server and set the preferences.

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_llexcel.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 top-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 5.5 tab, Definitions group, under Parameters , click Filter .

This imports the model parameters to the worksheet and creates a direct link to the Parameters node in the COMSOL model. Only the model parameters containing the prefix `extern_` are imported. It can be useful to filter parameters this way if a model contains many parameters and only a few should be available to Excel users. The link between the cell range in the worksheet and the COMSOL model is represented by the comment in the cell A4.

	A	B	C	D	E	F
1	COMSOL Model					
2	Filename:	Multiphysics\applications\LiveLink_for_Excel\Tutor				
3						
4	**Parameter**	**Expressions**	**Unit**	**Description**	**Value**	**Unit**
5	extern_L	9	cm	Length of busbar	0,09	m
6	extern_Vtot	20	mV	Voltage	0,02	V

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 `extern_L` parameter to 5 cm:

3 In cell B5 enter `5`, the parameter `extern_L` is then automatically updated to 5 in COMSOL Multiphysics.

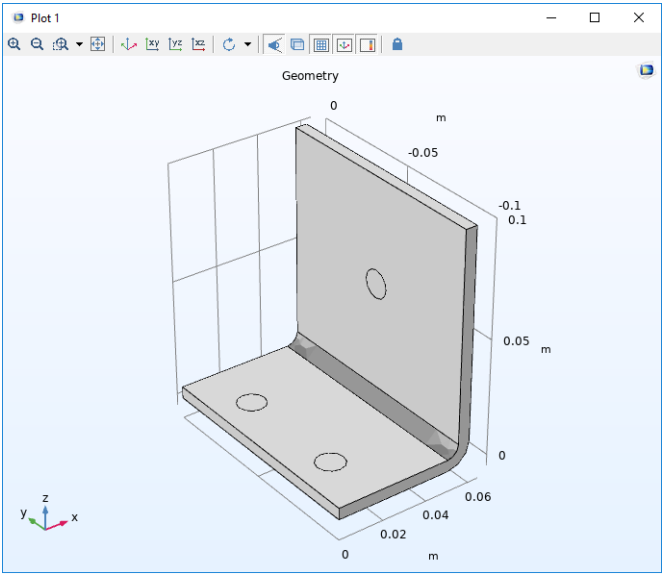
Note: If you prefer a manual parameter update, go to the COMSOL preferences in the Backstage view and clear Automatic update check box. Then you can use the update button to update the Parameters node linked to the selected cell by the comment. You can update several model definitions, such as parameters,

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:

4 In the View group, click the Geometry button .

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



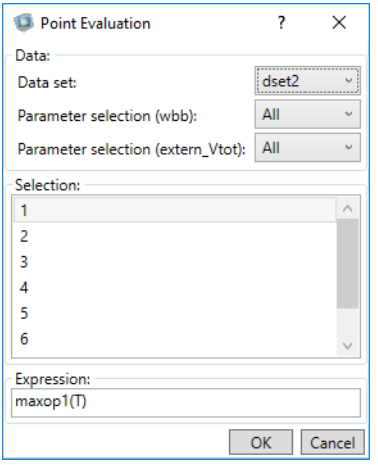
5 You will now set the value of parameter `extern_L` back to 9 cm, in cell B5 enter `9` and press Enter.

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 dset 2 in the Data set menu list. The solution data set 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:	Multiphysics\applications\LiveLink_for_Excel\Tutorials\busbar_llexcel.mph								
3										
4	**Parameter**	**Expressions**	**Unit**	**Description**	**Value**	**Unit**		**wbb (m)**	**extern_Vtot (V)**	**Maximum 1 (K), Point: 1**
5	extern_L	5	cm	Length of busbar	0.05	m		0.1	0.005	294.0776715
6	extern_Vtot	20	mV	Voltage	0.02	V		0.1	0.01	296.3624144
7								0.1	0.02	303.1640932
8								0.1	0.03	313.7664051
9								0.1	0.04	327.4436733

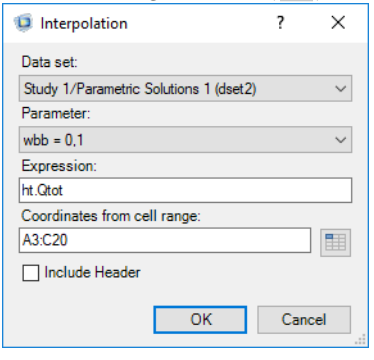
INTERPOLATION

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			
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

4. In the Data set 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	0.000285209	0.001140966	0.004563865	0.010268697	0.018255462
4	2.50E-02	0	0	82.16153326	328.646133	1314.584532	2957.815197	5258.338129
5	5.00E-02	0	0	117.5928242	470.3712969	1881.485187	4233.341672	7525.94075
6	0	-1.25E-02	0	18.24137619	72.96550475	291.862019	656.6895428	1167.448076
7	2.50E-02	-1.25E-02	0	118.2585361	473.0341446	1892.136578	4257.307301	7568.546313
8	5.00E-02	-1.25E-02	0	119.7233991	478.8935964	1915.574386	4310.042368	7662.297543
9	0	-2.50E-02	0	0.010070478	0.040281913	0.161127651	0.362537215	0.644510605
10	2.50E-02	-2.50E-02	0	200.9120425	803.6481699	3214.592679	7232.833529	12858.37072
11	5.00E-02	-2.50E-02	0	122.357399	489.4295962	1957.718385	4404.866366	7830.873539
12	0	0	5.00E-03	0.000237859	0.000951435	0.003805739	0.008562913	0.015222957
13	2.50E-02	0	5.00E-03	81.56887876	326.275515	1305.10206	2936.479635	5220.40824
14	5.00E-02	0	5.00E-03	117.5857422	470.3429689	1881.371876	4233.08672	7525.487503
15	0	-1.25E-02	5.00E-03	17.63350982	70.53403929	282.1361572	634.8063537	1128.544629
16	2.50E-02	-1.25E-02	5.00E-03	118.3439561	473.3758242	1893.503297	4260.382418	7574.013187
17	5.00E-02	-1.25E-02	5.00E-03	119.7711525	479.0846099	1916.33844	4311.761489	7665.353759
18	0	-2.50E-02	5.00E-03	0.047673835	0.19069534	0.762781361	1.716258062	3.051125443
19	2.50E-02	-2.50E-02	5.00E-03	154.6237111	618.4948443	2473.979377	5566.453599	9895.917509
20	5.00E-02	-2.50E-02	5.00E-03	122.3601013	489.4404051	1957.76162	4404.963646	7831.046481

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.

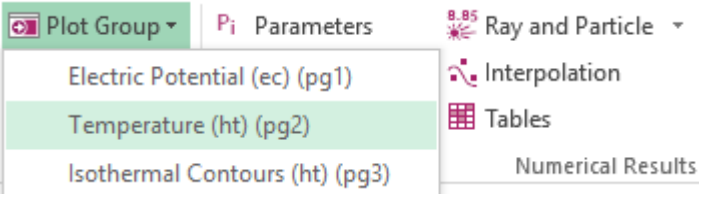
8 You can format the worksheet for instance include column labels, insert cell at your convenience. Make sure that the cell comment remain in the worksheet to allow future update.

	A	B	C	D	E	F	G	H
1				Qtot[W] (wbb = 0.05m)				
2	x	y	z	Vtot = 5mV	Vtot = 10mV	Vtot = 20mV	Vtot = 30mV	Vtot = 40mV
3	0	0	0	0.117269361	0.469077442	1.876309769	4.22169698	7.505239075
4	2.50E-02	0	0	477.5731277	1910.292511	7641.170043	17192.6326	30564.68017
5	5.00E-02	0	0	451.6437161	1806.574865	7226.299458	16259.17378	28905.19783
6	0	-1.25E-02	0	0.007954629	0.031818517	0.127274067	0.286366651	0.509096269
7	2.50E-02	-1.25E-02	0	433.0234795	1732.093918	6928.375672	15588.84526	27713.50269
8	5.00E-02	-1.25E-02	0	451.555438	1806.221752	7224.887008	16255.99577	28899.54803
9	0	-2.50E-02	0	0.019301632	0.077206529	0.308826115	0.694858758	1.235304459
10	2.50E-02	-2.50E-02	0	478.8569365	1915.427746	7661.710984	17238.84971	30646.84394
11	5.00E-02	-2.50E-02	0	451.7749545	1807.099818	7228.399272	16263.89836	28913.59709
12	0	0	5.00E-03	0.168721302	0.674885209	2.699540837	6.073966883	10.79816335
13	2.50E-02	0	5.00E-03	478.4776397	1913.910559	7655.642236	17225.19503	30622.56894
14	5.00E-02	0	5.00E-03	451.6429691	1806.571877	7226.287506	16259.14689	28905.15002
15	0	-1.25E-02	5.00E-03	0.026275891	0.105103564	0.420414256	0.945932076	1.681657024
16	2.50E-02	-1.25E-02	5.00E-03	363.9984622	1455.993849	5823.975396	13103.94464	23295.90158
17	5.00E-02	-1.25E-02	5.00E-03	451.5523019	1806.209207	7224.83683	16255.88287	28899.34732
18	0	-2.50E-02	5.00E-03	0.016769607	0.067078429	0.268313715	0.603705859	1.073254861
19	2.50E-02	-2.50E-02	5.00E-03	477.8077881	1911.231152	7644.924609	17201.08037	30579.69844
20	5.00E-02	-2.50E-02	5.00E-03	451.7644029	1807.057612	7228.230447	16263.51851	28912.92179

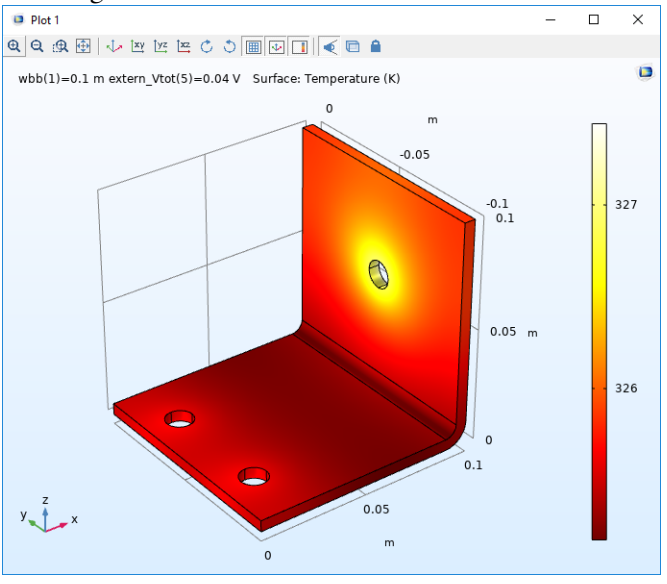
Displaying the Solution

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) (pg2).



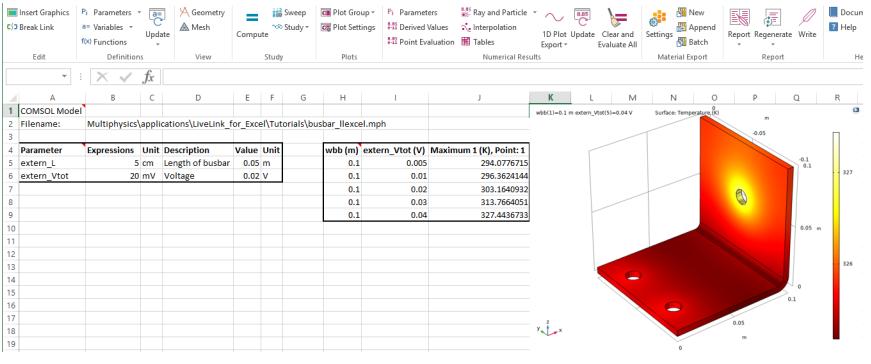
The graphics window now displays the temperature distribution in the busbar as in the figure below:



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.

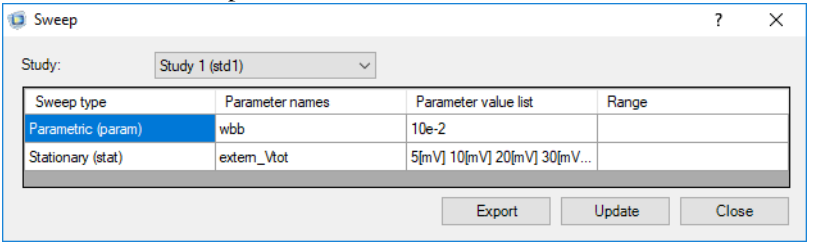
To insert the displayed image into the worksheet, in Sheet 1 select cell K1, then click Insert Graphics from the Edit group.



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.

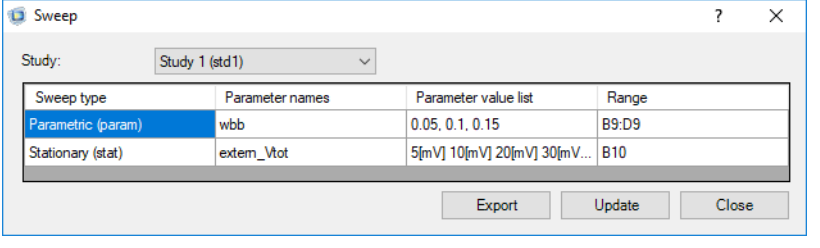


2 Click Export to export the sweep parameter data to the worksheet and create a link between the selected cell range and the model.

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.

1 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.

1. 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.

2. 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:	Multiphysics\applications\LiveLink_for_Excel\Tutorials\busbar_llexcel.mph								
3										
4	**Parameter**	**Expressions**	**Unit**	**Description**	**Value**	**Unit**		**wbb (m)**	**extern_Vtot (V)**	**Maximum 1 (K), Point: 1**
5	extern_L	5	cm	Length of busbar	0.05	m		0.05	0.005	294.7634785
6	extern_Vtot	20	mV	Voltage	0.02	V		0.05	0.01	298.3921782
7								0.05	0.02	310.0511009
8	**Name**	**Expressions**						0.05	0.03	327.7515729
9	wbb	5.00E-02	0.1	0.15				0.05	0.04	350.2577614
10	extern_Vtot	5[mV] 10[mV]	20[mV]	30[mV] 40[mV]				0.1	0.005	294.0776021
11								0.1	0.01	296.3621594
12								0.1	0.02	303.16315
13								0.1	0.03	313.7642659
14								0.1	0.04	327.4399389
15								0.15	0.005	293.8077463
16								0.15	0.01	295.5113014
17								0.15	0.02	300.6520318
18								0.15	0.03	308.2181707
19								0.15	0.04	318.3606863

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] (wbb = 0.05m)				
2	x	y	z	Vtot = 5mV	Vtot = 10mV	Vtot = 20mV	Vtot = 30mV	Vtot = 40mV
3	0	0	0	0.117269361	0.469077442	1.876309769	4.22169698	7.505239075
4	2.50E-02	0	0	477.5731277	1910.292511	7641.170043	17192.6326	30564.68017
5	5.00E-02	0	0	451.6437161	1806.574865	7226.299458	16259.17378	28905.19783
6	0	-1.25E-02	0	0.007954629	0.031818517	0.127274067	0.286366651	0.509096269
7	2.50E-02	-1.25E-02	0	433.0234795	1732.093918	6928.375672	15588.84526	27713.50269
8	5.00E-02	-1.25E-02	0	451.555438	1806.221752	7224.887008	16255.99577	28899.54803
9	0	-2.50E-02	0	0.019301632	0.077206529	0.308826115	0.694858758	1.235304459
10	2.50E-02	-2.50E-02	0	478.8569365	1915.427746	7661.710984	17238.84971	30646.84394
11	5.00E-02	-2.50E-02	0	451.7749545	1807.099818	7228.399272	16263.89836	28913.59709
12	0	0	5.00E-03	0.168721302	0.674885209	2.699540837	6.073966883	10.79816335
13	2.50E-02	0	5.00E-03	478.4776397	1913.910559	7655.642236	17225.19503	30622.56894
14	5.00E-02	0	5.00E-03	451.6429691	1806.571877	7226.287506	16259.14689	28905.15002
15	0	-1.25E-02	5.00E-03	0.026275891	0.105103564	0.420414256	0.945932076	1.681657024
16	2.50E-02	-1.25E-02	5.00E-03	363.9984622	1455.993849	5823.975396	13103.94464	23295.90158
17	5.00E-02	-1.25E-02	5.00E-03	451.5523019	1806.209207	7224.83683	16255.88287	28899.34732
18	0	-2.50E-02	5.00E-03	0.016769607	0.067078429	0.268313715	0.603705859	1.073254861
19	2.50E-02	-2.50E-02	5.00E-03	477.8077881	1911.231152	7644.924609	17201.08037	30579.69844
20	5.00E-02	-2.50E-02	5.00E-03	451.7644029	1807.057612	7228.230447	16263.51851	28912.92179

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.

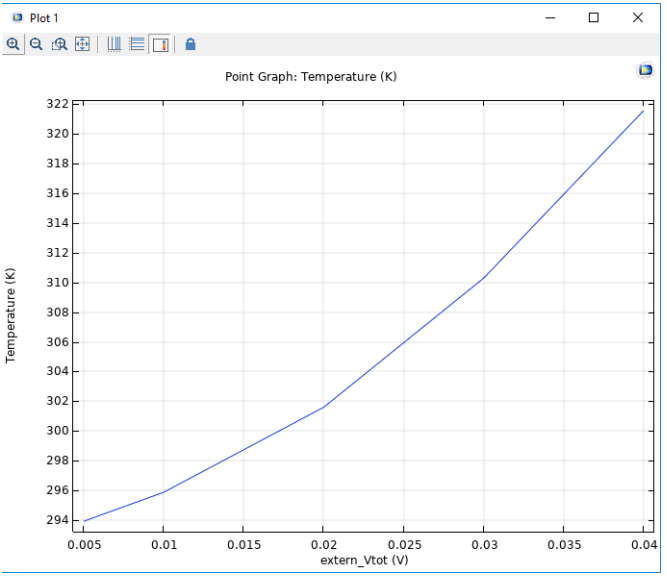
	A	B	C	D	E	F	G	H	I	J	K	L	M
1				Qtot[W] (wbb = 0.05m)					Qtot[W] (wbb = 0.1m)				
2	x	y	z	Vtot = 5mV	Vtot = 10mV	Vtot = 20mV	Vtot = 30mV	Vtot = 40mV	Vtot = 5mV	Vtot = 10mV	Vtot = 20mV	Vtot = 30mV	Vtot = 40mV
3	0	0	0	0.117269361	0.469077442	1.876309769	4.22169698	7.505239075	0.000285209	0.001140835	0.004563342	0.010267519	0.018253368
4	2.50E-02	0	0	477.5731277	1910.292511	7641.170043	17192.6326	30564.68017	82.16583858	328.6633543	1314.653417	2957.970189	5258.613669
5	5.00E-02	0	0	451.6437161	1806.574865	7226.299458	16259.17378	28905.19783	117.5886557	470.3546228	1881.418491	4233.191605	7525.673964
6	0	-1.25E-02	0	0.007954629	0.031818517	0.127274067	0.286366651	0.509096269	18.24019312	72.96077246	291.8430899	656.6469522	1167.372359
7	2.50E-02	-1.25E-02	0	433.0234795	1732.093918	6928.375672	15588.84526	27713.50269	118.2613415	473.0453661	1892.181464	4257.408295	7568.725857
8	5.00E-02	-1.25E-02	0	451.555438	1806.221752	7224.887008	16255.99577	28899.54803	119.7184615	478.8738458	1915.495383	4309.864612	7661.981533
9	0	-2.50E-02	0	0.019301632	0.077206529	0.308826115	0.694858758	1.235304459	0.010236979	0.040947915	0.163791659	0.368531233	0.655166637
10	2.50E-02	-2.50E-02	0	478.8569365	1915.427746	7661.710984	17238.84971	30646.84394	200.9196425	803.6785698	3214.714279	7233.107129	12858.85712
11	5.00E-02	-2.50E-02	0	451.7749545	1807.099818	7228.399272	16263.89836	28913.59709	122.3674671	489.4698683	1957.879473	4405.228814	7831.517892
12	0	0	5.00E-03	0.168721302	0.674885209	2.699540837	6.073966883	10.79816335	0.000237858	0.000951433	0.003805733	0.008562898	0.01522293
13	2.50E-02	0	5.00E-03	478.4776397	1913.910559	7655.642236	17225.19503	30622.56894	81.57262683	326.2905073	1305.162029	2936.614566	5220.648117
14	5.00E-02	0	5.00E-03	451.6429691	1806.571877	7226.287506	16259.14689	28905.15002	117.581578	470.3263119	1881.305248	4232.936807	7525.22099
15	0	-1.25E-02	5.00E-03	0.026275891	0.105103564	0.420414256	0.945932076	1.681657024	17.63211434	70.52845737	282.1138295	634.7561163	1128.455318
16	2.50E-02	-1.25E-02	5.00E-03	363.9984622	1455.993849	5823.975396	13103.94464	23295.90158	118.3580992	473.4323969	1893.729587	4260.891572	7574.91835
17	5.00E-02	-1.25E-02	5.00E-03	451.5523019	1806.209207	7224.83683	16255.88287	28899.34732	119.7660976	479.0643906	1916.257562	4311.579515	7665.030249
18	0	-2.50E-02	5.00E-03	0.016769607	0.067078429	0.268313715	0.603705859	1.073254861	0.04755631	0.19022524	0.760900962	1.712027163	3.043603846
19	2.50E-02	-2.50E-02	5.00E-03	477.8077881	1911.231152	7644.924609	17201.08037	30579.69844	154.6216138	618.4864553	2473.945821	5566.378098	9895.783285
20	5.00E-02	-2.50E-02	5.00E-03	451.7644029	1807.057612	7228.230447	16263.51851	28912.92179	122.329644	489.3185762	1957.274305	4403.867186	7829.097219

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:

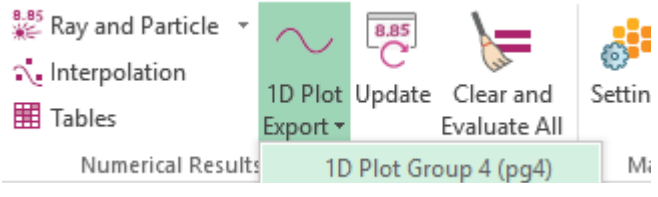


This plot can be inserted into Excel as a static image. Instead we will insert the data that makes up the plot and create a scatter chart in the Excel sheet.

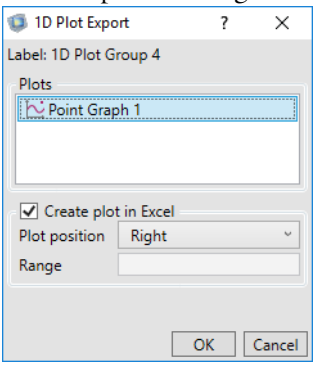
1 Add a new sheet.

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).

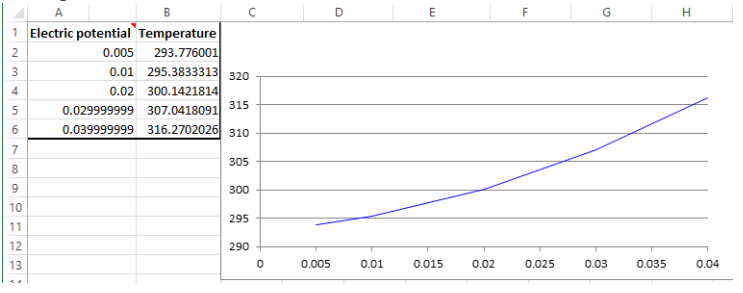
3 On the COMSOL ribbon tab, locate the 1D Plot Export button. Click on it see what 1D plots are available in the model and select 1D Plot Group 4 (pg4).



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:

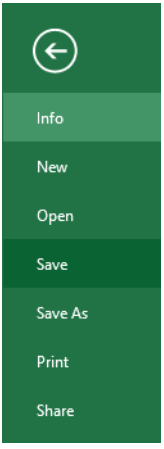


Saving the Model and the Workbook

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.

1 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_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

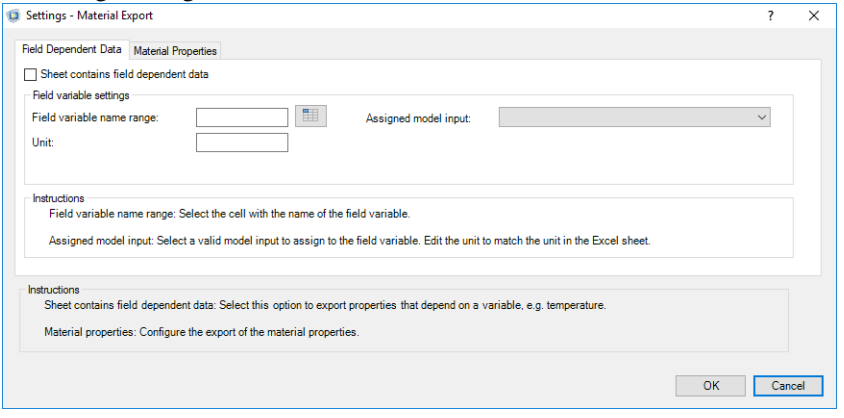
1 Start Excel and open the file `busbar_llexcel_data.xlsx` that you can find in your COMSOL Multiphysics installation directory and in the folder `applications/LiveLink_for_Excel/Tutorials`.

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^3.

	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

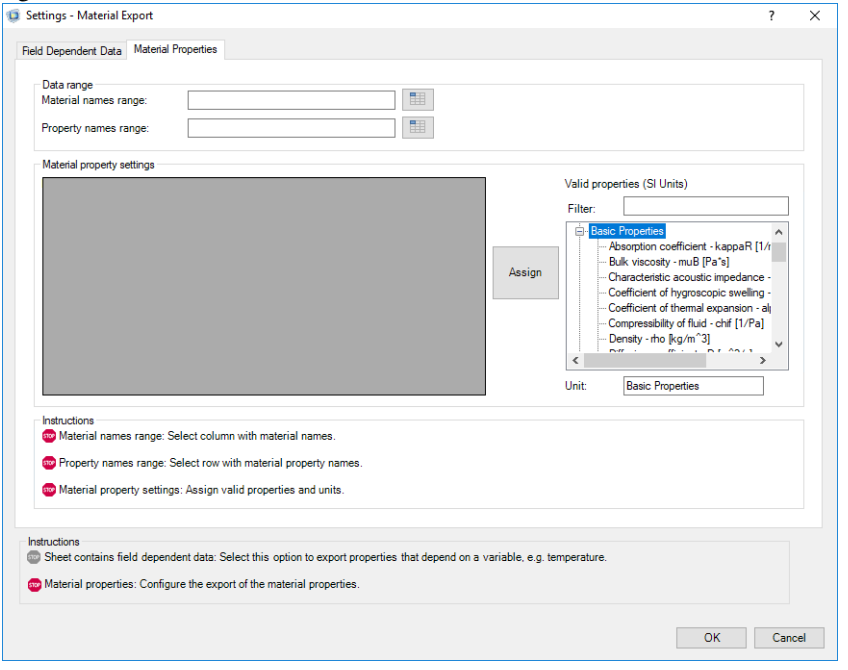
3 On the COMSOL 5.5 tab, click Settings to open the Material Export Settings dialog box.



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.

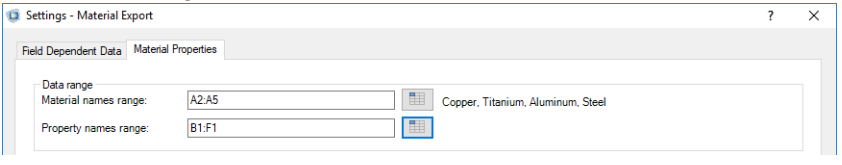


5 You will now select the cells containing the material names. Click the Range button ( ) next to the Material names range text field.

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.

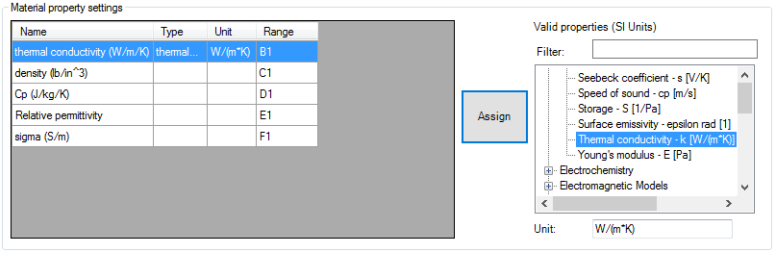
8 Select the range B1:F1 and click OK.



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 [W/(m*K)] and click Assign.

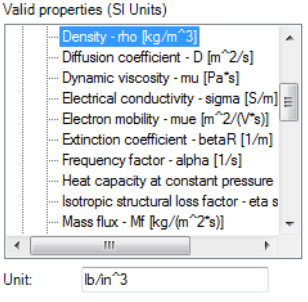


Note that the number of properties remaining to be assigned is displayed in the Instructions section.

11 Now select density (lb/in^3) from the Name column, then select Density [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.

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

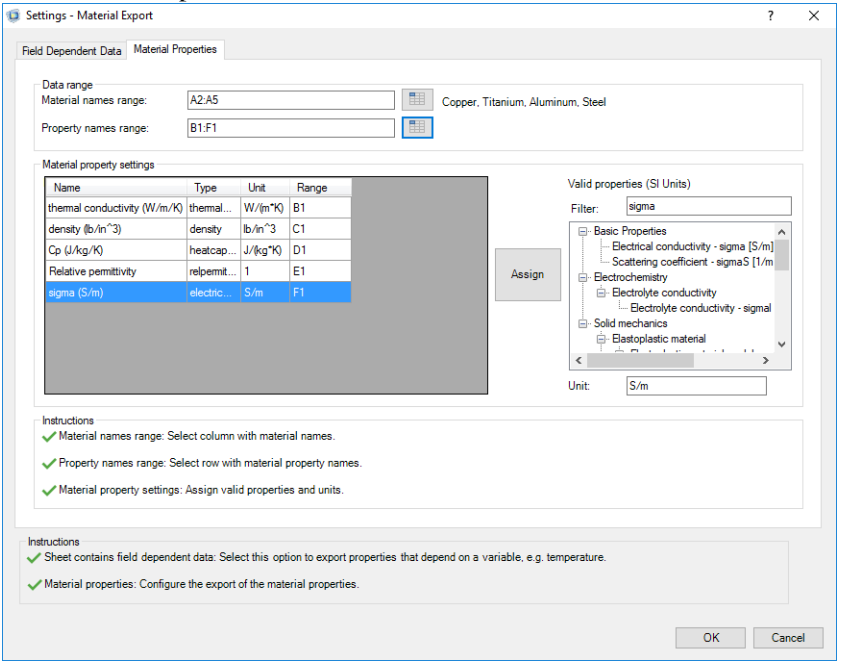


13 Click Assign to assign the properties and the unit to the data in the spreadsheet.

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 [J/(kg*K)]`	`J/(kg*K)`
`Relative permittivity`	`Relative permittivity [1]`	`1`
`sigma (S/m)`	`Electrical conductivity [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^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_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.

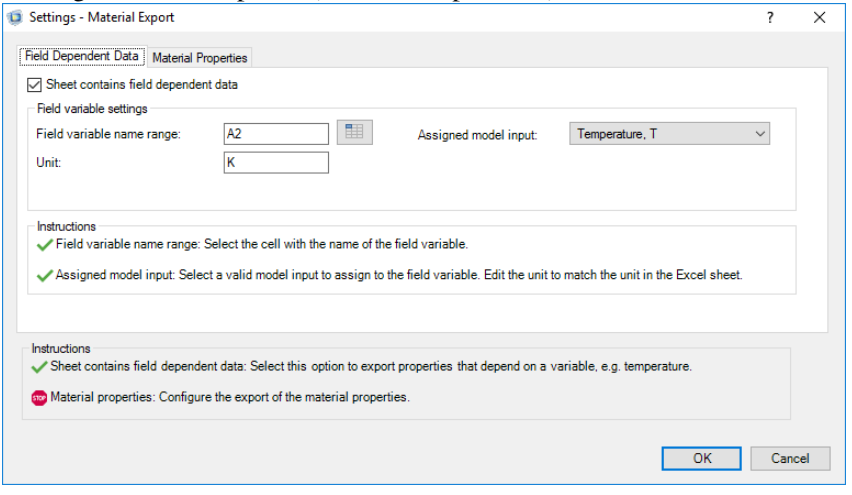
	A	B	C	D	E
1	Water, liquid				
2	T (K)	eta (Pa*s)	Cp (J/(kg*K))	rho (kg/m^3)	k (W/(m*K))
3	273.15	0.0017915	4216.2779	1003.9209	0.55623
4	278.15	0.0015356	4206.8543	1003.0467	0.56642
5	283.15	0.0013248	4198.8844	1002.0375	0.57614
6	288.15	0.0011516	4192.27	1000.8935	0.58541
7	293.15	0.0010093	4186.9181	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 5.5 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 ( ) next to the Material names range text field.

8 Select cell A1 in the sheet and click OK.

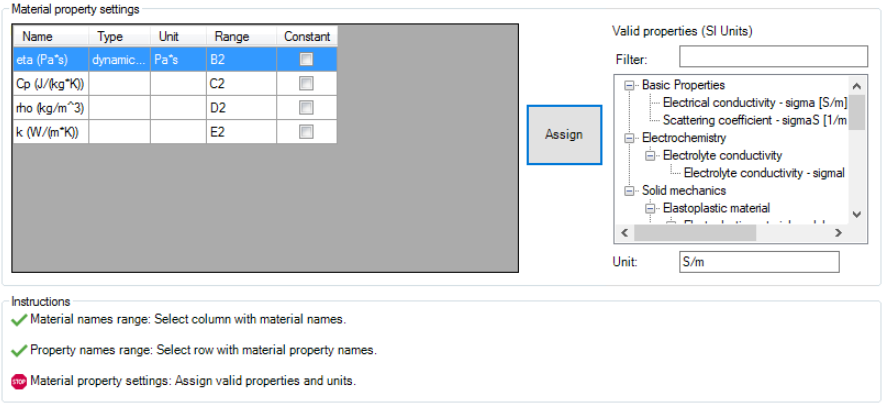
9 Continue by selecting the cells with the property names. Click the Range button ( ) to the right of the Property names range text field.

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 [Pa*s] and click Assign.

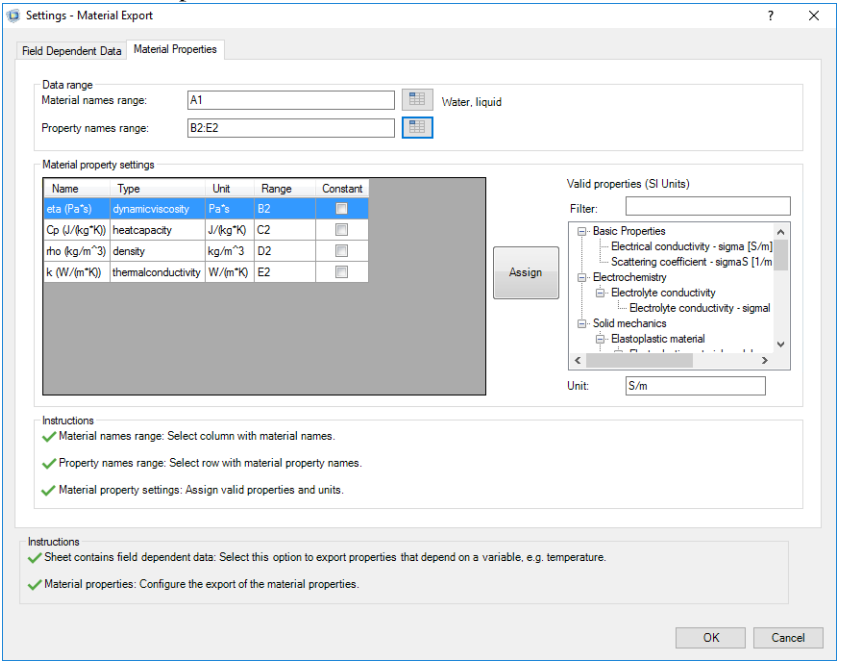


Note that the number of properties remaining to be assigned is displayed in the Instructions section.

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 [J/(kg*K)]`	`J/(kg*K)`
`rho(kg/m^3)`	`Density [kg/m^3]`	`kg/m^3`
`k (W/(m*K))`	`Thermal conductivity [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.

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 New 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/v55/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/v55/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 .

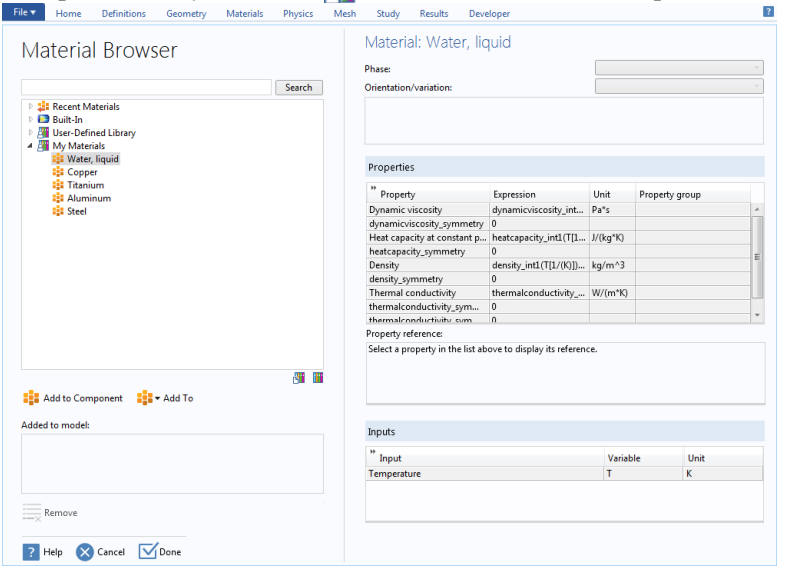
3 For the space dimension click 3D .

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.

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.

Settings

Parameters

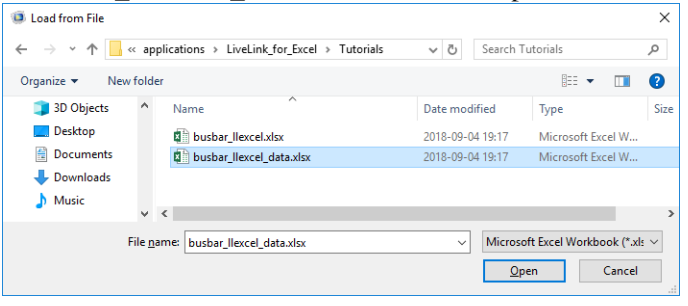
Label: Parameters 1

Parameters

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/(m²·K)	Heat transfer coefficient
Vtot	20[mV]	0.02 V	Applied voltage

2 To import parameters from an Excel file to the list, click Load from File located below the table.

3 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.



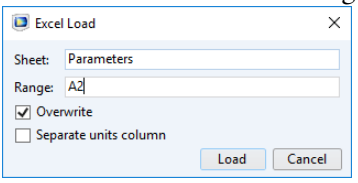
This opens the Excel Load dialog box.

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

	A	B	C
1	Name	Expression	Description
2	extern_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	extern_Vtot	20[mV]	Voltage

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

4 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` in the Range text field. Note that the range should not include the column headers.



Note: Another way to set which cell to import is to specify the cell range. In the case above it was possible to enter `A2;A5;A7` in the Range text field to import only the parameters `extern_L`, `wbb`, and `extern_Vtot`.

5 Click Load.

The parameter list is now imported to the model.

The two new parameters are added to the end of the table. Since the Overwrite option was selected for the import, the existing parameters are automatically updated with the data from the Excel file; thus the width parameter, `wbb`, has a new value of 10 cm.

Settings
Parameters

Parameters

Name	Expression	Value	Description
L	9[cm]	0.09 m	Length
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
htc	5[W/m^2/K]	5 W/(m²·K)	Heat transfer coefficient
Vtot	20[mV]	0.02 V	Applied voltage
extern_L	9[cm]	0.09 m	Length of busbar
extern_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.

You may notice that some parameters have the suffix `extern`; these correspond to the parameters that you can filter and link with a spreadsheet when running a model from within Excel. See the section Retrieving and Updating Model Parameters.

6 Replace the expression for the parameters L and Vtot according to the table below:

NAMES	EXPRESSION	VALUE	DESCRIPTION
L	extern_L	0.09 m	Length
Vtot	extern_Vtot	0.02 V	Applied voltage

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 f(x) 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.

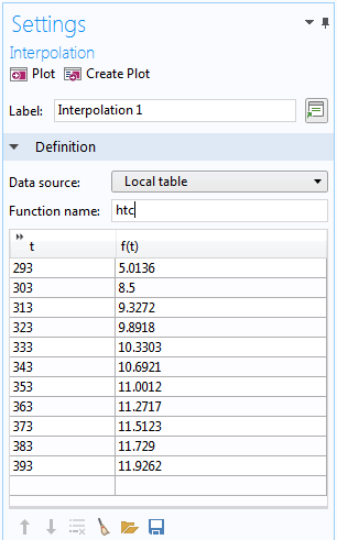
	A	B
1	T[K]	htc[W/m^2/K]
2	293	5.0136
3	303	8.5
4	313	9.3272
5	323	9.8918
6	333	10.3303
7	343	10.6921
8	353	11.0012
9	363	11.2717
10	373	11.5123
11	383	11.729
12	393	11.9262

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.

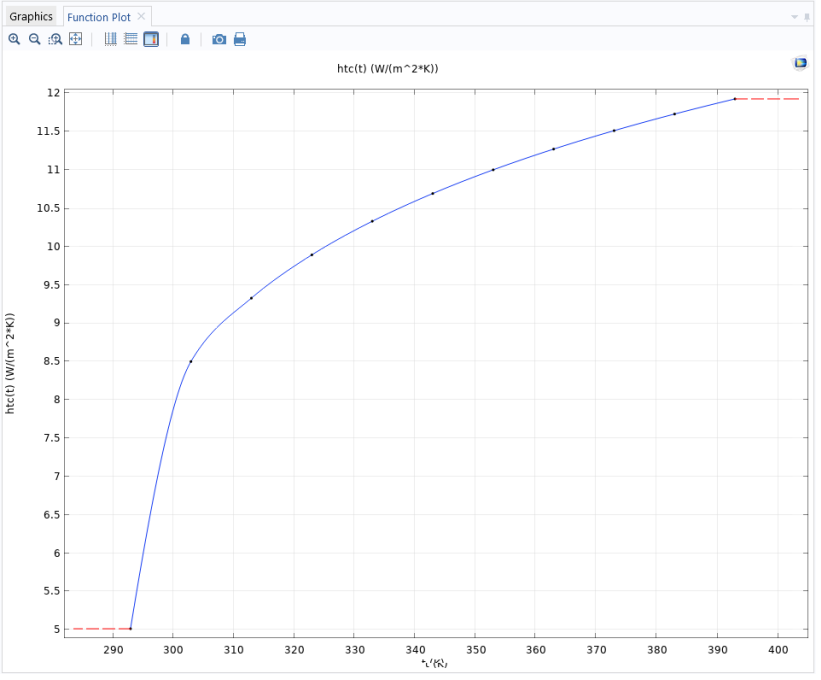
The interpolation table is filled using the data stored in the Excel file.



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 `extern_Vtot`, 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.

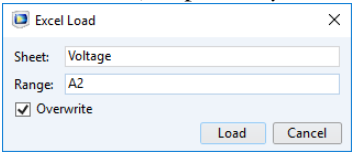
3 Under the Study Extensions section, select Auxiliary sweep and then click the Load from File button.

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_llexcel_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	extern_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.

1 On the Study toolbar, click Compute .

2 To view the temperature distribution in the busbar only, you need to add a selection to the solution data set. Under Results > Data Sets, right-click Study 1/Solution 1 and select Selection.

3 In the Settings window for Selection set the Geometric entity level to Domain, and select Domain 1.

4 To see the temperature distribution in the busbar for the last parameter value (40 mV), expand the Temperature (ht) node. In the Surface node settings windows, expands the Range section and clear Manual color range.