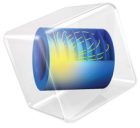


INTRODUCTION TO LiveLink™ *for* Excel®

5.3

COMSOL

Introduction to LiveLink™ *for* Excel®

Version: COMSOL 5.3

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. Moreover, LiveLink™ adds the capability to create a COMSOL® material library from data stored in a worksheet and brings it into COMSOL Multiphysics, while also enabling support for loading Excel files for parameter and variable lists in the COMSOL Desktop®.

Working with COMSOL® Models in Excel®

LiveLink™ 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

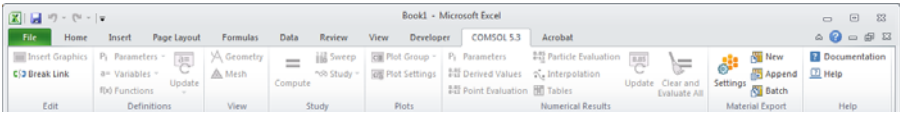
The example is based on a modified version of the model, Electrical Heating in a Busbar, from the COMSOL Multiphysics application library. The model analyzes the resistive heating of a busbar designed to conduct direct current; for details, see the *Introduction to COMSOL Multiphysics*.

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

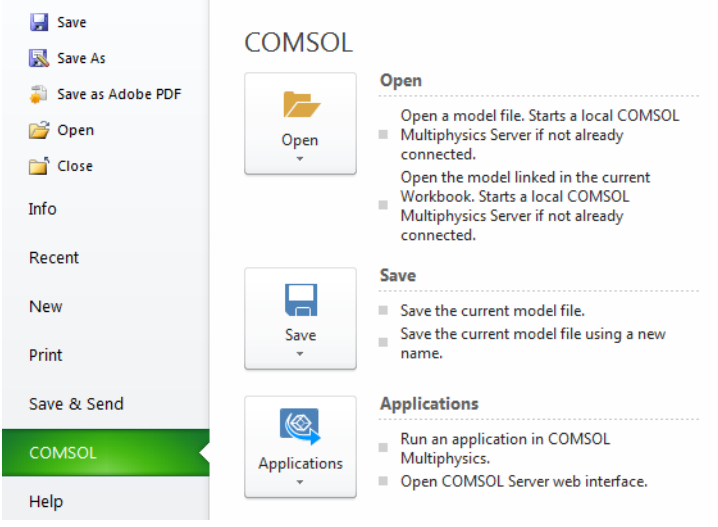
1 Start Excel.

2 In the Excel ribbon locate the COMSOL 5.3 tab.



Start with loading the model.

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



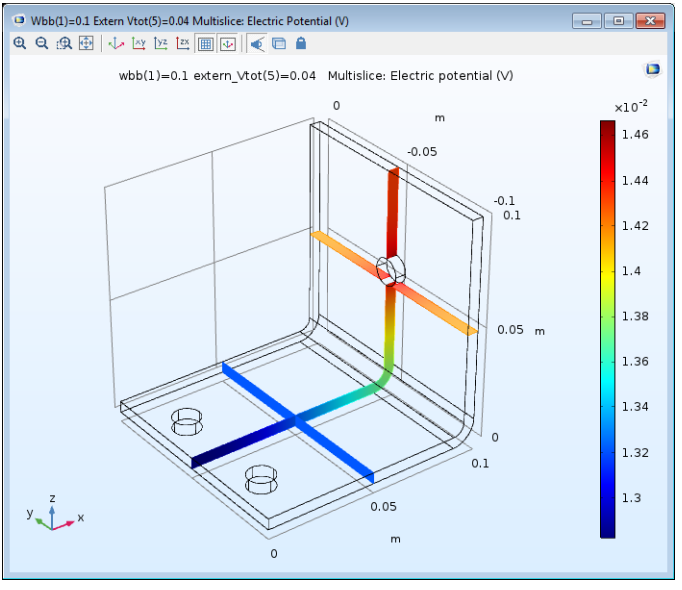
Note: In the COMSOL backstage view you can manage the COMSOL Multiphysics model, launch COMSOL Apps, connect manually to a server and set the preferences.

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
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	C
1	COMSOL Model		
2	Filename:	\Multiphysics\appli	
3			

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

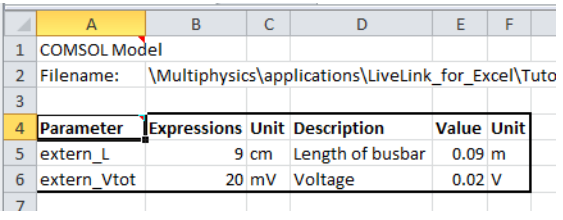
Retrieving and Updating Model Parameters

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

1 Select cell A4.

2 From the COMSOL 5.3 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. 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\Tuto				
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
7						

Note: You may retrieve parameters from 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.

You will now change the value of the `extern_L` parameter to 5 cm and update the model:

3 In cell B5 enter `5`.

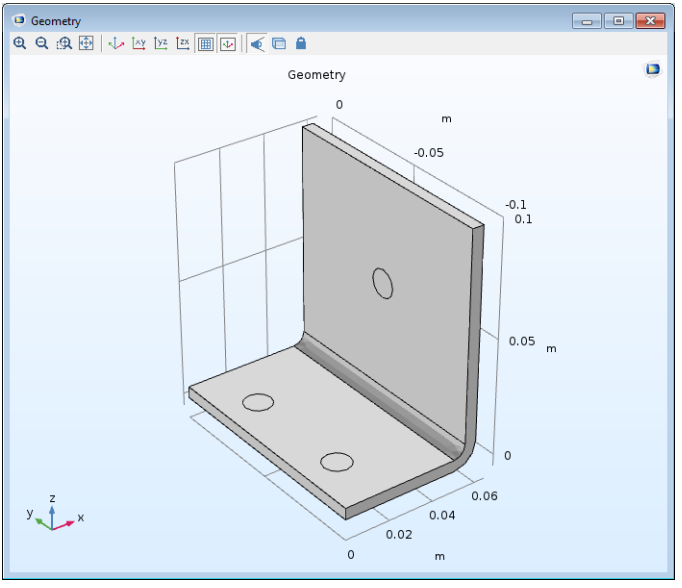
4 Select cell A4, then click the Update button , located in the Definitions group of the COMSOL tab.

Note: In the COMSOL Multiphysics model, this action only updates 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:

5 In the View group, click the Geometry button .

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



6 You will now set the value of parameter `extern_L` back to 9 cm, in cell B5 enter 9 and press Enter. Under Update button , click Update All to update all model definitions in the worksheet without selecting a cell that contains a comment.

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.

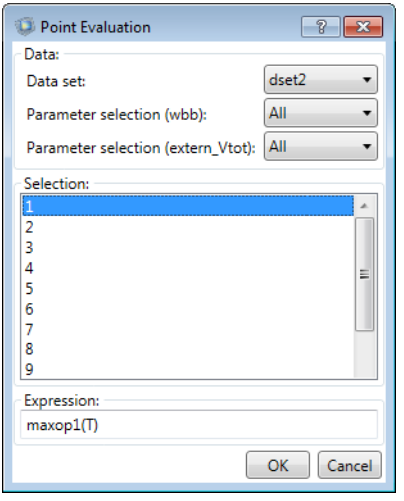
Point Evaluation

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**	**extern_Vtot (V)**	**Maximum 1 (K), Point: 1**
5	extern_L	9	cm	Length of busbar	0.09	m		0.1	0.005	294.0711192
6	extern_Vtot	20	mV	Voltage	0.02	V		0.1	0.01	296.3428413
7								0.1	0.02	303.1063965
8								0.1	0.03	313.6437185
9								0.1	0.04	327.2472832
10										

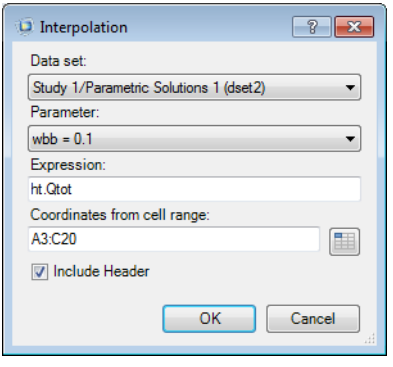
INTERPOLATION

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

1. In the worksheet Sheet2, enter the coordinates as shown in the figure to the right.
2. 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

3. In the Data set menu list, select Study 1/ Parametric Solutions 1(dset2).
4. In the Interpolation dialog box, in the Expression text field, enter `ht.Qtot`, which is the total heat source to be evaluated.
5. 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.
6. Select Include Header.



7 Finally click OK.

	A	B	C	D	E	F	G	H
1								
2	x	y	z	ht.Qtot (A3:C20)				
3	0	0	0	4.42424E-05	0.000176969	0.000707878	0.001592725	0.002831512
4	2.50E-02	0	0	79.71271006	318.8508402	1275.403361	2869.657562	5101.613444
5	5.00E-02	0	0	115.609924	462.439696	1849.758784	4161.957264	7399.035137
6	0	-1.25E-02	0	17.8803299	71.52131959	286.0852784	643.6918763	1144.341113
7	2.50E-02	-1.25E-02	0	114.1394006	456.5576026	1826.23041	4109.018423	7304.921642
8	5.00E-02	-1.25E-02	0	117.9038766	471.6155064	1886.462026	4244.539558	7545.848103
9	0	-2.50E-02	0	0.045573518	0.182294072	0.729176286	1.640646645	2.916705146
10	2.50E-02	-2.50E-02	0	215.9265584	863.7062335	3454.824934	7773.356102	13819.29974
11	5.00E-02	-2.50E-02	0	120.2674982	481.0699929	1924.279972	4329.629936	7697.119887
12	0	0	5.00E-03	0.000154889	0.000619557	0.002478229	0.005576016	0.009912917
13	2.50E-02	0	5.00E-03	79.68594285	318.7437714	1274.975086	2868.693943	5099.900343
14	5.00E-02	0	5.00E-03	115.6016158	462.4064633	1849.625853	4161.65817	7398.503413
15	0	-1.25E-02	5.00E-03	17.42932718	69.71730873	278.8692349	627.4557786	1115.47694
16	2.50E-02	-1.25E-02	5.00E-03	113.8470178	455.3880711	1821.552284	4098.49264	7286.209138
17	5.00E-02	-1.25E-02	5.00E-03	117.8614222	471.4456888	1885.782755	4243.011199	7543.131021
18	0	-2.50E-02	5.00E-03	0.010764435	0.043057741	0.172230962	0.387519665	0.688923849
19	2.50E-02	-2.50E-02	5.00E-03	147.5785277	590.314111	2361.256444	5312.826999	9445.025776
20	5.00E-02	-2.50E-02	5.00E-03	120.2788416	481.1153663	1924.461465	4330.038297	7697.845861

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.

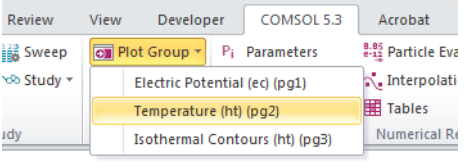
8 You can format the worksheet for instance include column labels, insert cell at your convenience. Make sure that the cell comment remain in the worksheet to allow future update.

	A	B	C	D	E	F	G	H
1				Qtot[W] (wbb= 0.1m)				
2	x	y	z	Vtot = 5mV	Vtot = 10mV	Vtot = 20mV	Vtot = 30mV	Vtot = 40mV
3	0	0	0	4.42424E-05	0.000176969	0.000707878	0.001592725	0.002831512
4	2.50E-02	0	0	79.71271006	318.8508402	1275.403361	2869.657562	5101.613444
5	5.00E-02	0	0	115.609924	462.439696	1849.758784	4161.957264	7399.035137
6	0	-1.25E-02	0	17.8803299	71.52131959	286.0852784	643.6918763	1144.341113
7	2.50E-02	-1.25E-02	0	114.1394006	456.5576026	1826.23041	4109.018423	7304.921642
8	5.00E-02	-1.25E-02	0	117.9038766	471.6155064	1886.462026	4244.539558	7545.848103
9	0	-2.50E-02	0	0.045573518	0.182294072	0.729176286	1.640646645	2.916705146
10	2.50E-02	-2.50E-02	0	215.9265584	863.7062335	3454.824934	7773.356102	13819.29974
11	5.00E-02	-2.50E-02	0	120.2674982	481.0699929	1924.279972	4329.629936	7697.119887
12	0	0	5.00E-03	0.000154889	0.000619557	0.002478229	0.005576016	0.009912917
13	2.50E-02	0	5.00E-03	79.68594285	318.7437714	1274.975086	2868.693943	5099.900343
14	5.00E-02	0	5.00E-03	115.6016158	462.4064633	1849.625853	4161.65817	7398.503413
15	0	-1.25E-02	5.00E-03	17.42932718	69.71730873	278.8692349	627.4557786	1115.47694
16	2.50E-02	-1.25E-02	5.00E-03	113.8470178	455.3880711	1821.552284	4098.49264	7286.209138
17	5.00E-02	-1.25E-02	5.00E-03	117.8614222	471.4456888	1885.782755	4243.011199	7543.131021
18	0	-2.50E-02	5.00E-03	0.010764435	0.043057741	0.172230962	0.387519665	0.688923849
19	2.50E-02	-2.50E-02	5.00E-03	147.5785277	590.314111	2361.256444	5312.826999	9445.025776
20	5.00E-02	-2.50E-02	5.00E-03	120.2788416	481.1153663	1924.461465	4330.038297	7697.845861

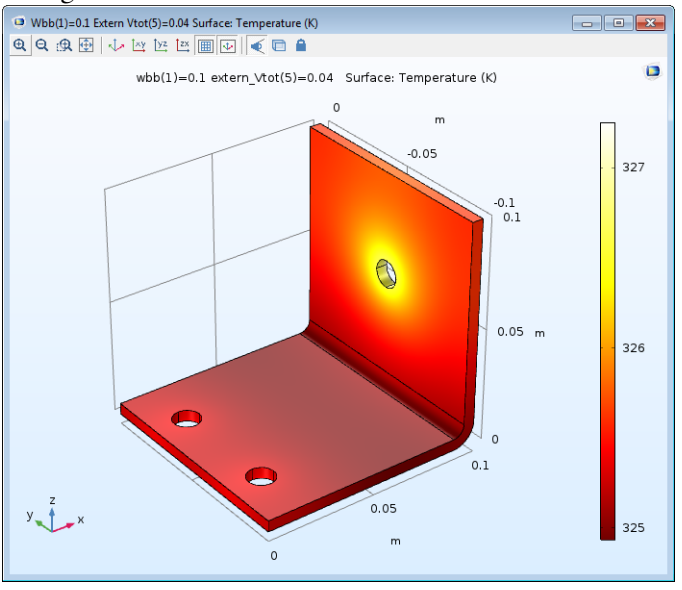
Displaying the Solution

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).



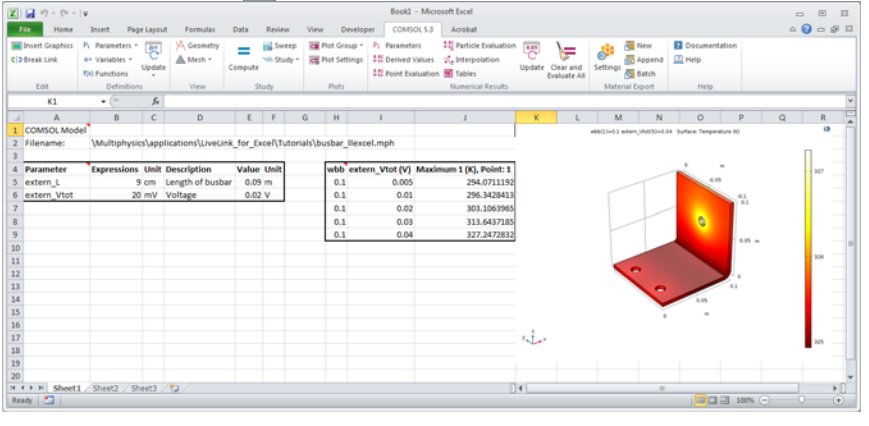
The graphics window now display the temperature distribution in the busbar as in the figure below:



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.

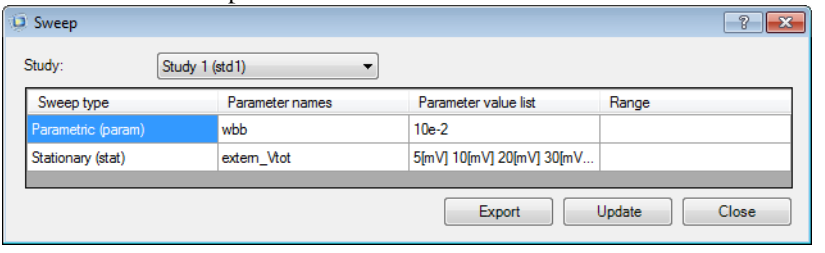
To insert the displayed image into the worksheet, in Sheet 1 select cell K1, then click Insert Graphics from the Edit group.



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.

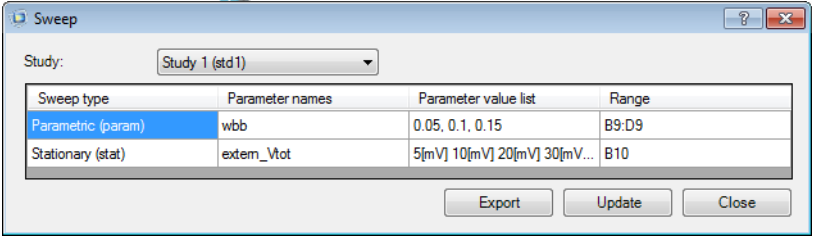


2 Click Export to export the sweep parameter data to the worksheet and create a link between the selected cell range and the model.

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**	**extern_Vtot (V)**	**Maximum 1 (K), Point: 1**
5	extern_L	9	cm	Length of busbar	0.09	m		0.05	0.005	294.7634749
6	extern_Vtot	20	mV	Voltage	0.02	V		0.05	0.01	298.3920938
7								0.05	0.02	310.0506599
8	**Name**	**Expressions**						0.05	0.03	327.7504179
9	wbb	5.00E-02	0.1	0.15				0.05	0.04	350.2555449
10	extern_Vtot	5[mV]	10[mV]	20[mV] 30[mV] 40[mV]				0.1	0.005	294.0784463
11								0.1	0.01	296.3648185
12								0.1	0.02	303.171561
13								0.1	0.03	313.7825084
14								0.1	0.04	327.4700566
15								0.15	0.005	293.8081225
16								0.15	0.01	295.5125207
17								0.15	0.02	300.6555413
18								0.15	0.03	308.2259005
19								0.15	0.04	318.3734007

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	0.121610671	0.486442686	1.945770743	4.377984171	7.78308297
4	2.50E-02	0	0	478.0639838	1912.255935	7649.02374	17210.30342	30596.09496
5	5.00E-02	0	0	451.713616	1806.854464	7227.417856	16261.69018	28909.67142
6	0	-1.25E-02	0	0.0085497	0.034198801	0.136795203	0.307789206	0.547180811
7	2.50E-02	-1.25E-02	0	453.7681952	1815.072781	7260.291123	16335.65503	29041.16449
8	5.00E-02	-1.25E-02	0	451.6142856	1806.457142	7225.82857	16258.11428	28903.31428
9	0	-2.50E-02	0	0.017918993	0.071675973	0.286703893	0.64508376	1.146815574
10	2.50E-02	-2.50E-02	0	480.141548	1920.566192	7682.264768	17285.09573	30729.05907
11	5.00E-02	-2.50E-02	0	451.8155935	1807.262374	7229.049496	16265.36137	28916.19799
12	0	0	5.00E-03	0.160269988	0.641079953	2.564319812	5.769719577	10.25727925
13	2.50E-02	0	5.00E-03	479.0510267	1916.204107	7664.816427	17245.83696	30659.26571
14	5.00E-02	0	5.00E-03	451.7157371	1806.862948	7227.451793	16261.76653	28909.80717
15	0	-1.25E-02	5.00E-03	0.027903349	0.111613397	0.446453586	1.004520569	1.785814345
16	2.50E-02	-1.25E-02	5.00E-03	363.9640818	1455.856327	5823.425309	13102.70695	23293.70124
17	5.00E-02	-1.25E-02	5.00E-03	451.6073387	1806.429355	7225.717419	16257.86419	28902.86967
18	0	-2.50E-02	5.00E-03	0.015632227	0.062528909	0.250115637	0.562760184	1.000462549
19	2.50E-02	-2.50E-02	5.00E-03	477.969502	1911.878008	7647.512032	17206.90207	30590.04813
20	5.00E-02	-2.50E-02	5.00E-03	451.8037029	1807.214811	7228.859246	16264.9333	28915.43698

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 and paste it ate 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.

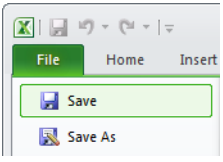
	A	B	C	D	E	F	G	H	I	J	K	L	M
1				Qtot[W] (wbb= 0.05m)					Qtot[W] (wbb= 0.1m)				
2	x	y	z	Vtot = 5mV	Vtot = 10mV	Vtot = 20mV	Vtot = 30mV	Vtot = 40mV	Vtot = 5mV	Vtot = 10mV	Vtot = 20mV	Vtot = 30mV	Vtot = 40mV
3	0	0	0	0.121610671	0.486442686	1.945770743	4.377984171	7.78308297	0.000273769	0.001095076	0.004380304	0.009855684	0.017521217
4	2.50E-02	0	0	478.0639838	1912.255935	7649.02374	17210.30342	30596.09496	82.38715782	329.5486313	1318.194525	2965.937681	5272.7781
5	5.00E-02	0	0	451.713616	1806.854464	7227.417856	16261.69018	28909.67142	117.8517546	471.4070184	1885.628074	4242.663166	7542.512295
6	0	-1.25E-02	0	0.0085497	0.034198801	0.136795203	0.307789206	0.547180811	18.34902588	73.39610352	293.5844141	660.5649317	1174.337656
7	2.50E-02	-1.25E-02	0	453.7681952	1815.072781	7260.291123	16335.65503	29041.16449	118.9871057	475.9484229	1903.793692	4283.535806	7615.174767
8	5.00E-02	-1.25E-02	0	451.6142856	1806.457142	7225.82857	16258.11428	28903.31428	119.9985817	479.9943267	1919.977307	4319.94894	7679.909227
9	0	-2.50E-02	0	0.017918993	0.071675973	0.286703893	0.64508376	1.146815574	0.012145696	0.048582784	0.194331135	0.437245055	0.777324542
10	2.50E-02	-2.50E-02	0	480.141548	1920.566192	7682.264768	17285.09573	30729.05907	206.9540036	827.8160146	3311.264058	7450.344131	13245.05623
11	5.00E-02	-2.50E-02	0	451.8155935	1807.262374	7229.049496	16265.36137	28916.19799	122.6022307	490.408923	1961.635692	4413.680307	7846.542767
12	0	0	5.00E-03	0.160269988	0.641079953	2.564319812	5.769719577	10.25727925	0.000226179	0.000904718	0.003618871	0.00814246	0.014475485
13	2.50E-02	0	5.00E-03	479.0510267	1916.204107	7664.816427	17245.83696	30659.26571	81.80950314	327.2380125	1308.95205	2945.142113	5235.808201
14	5.00E-02	0	5.00E-03	451.7157371	1806.862948	7227.451793	16261.76653	28909.80717	117.8220288	471.2881151	1885.15246	4241.593036	7540.609841
15	0	-1.25E-02	5.00E-03	0.027903349	0.111613397	0.446453586	1.004520569	1.785814345	17.81110244	71.24440974	284.977639	641.1996877	1139.910556
16	2.50E-02	-1.25E-02	5.00E-03	363.9640818	1455.856327	5823.425309	13102.70695	23293.70124	118.3421426	473.3685704	1893.474282	4260.317133	7573.897126
17	5.00E-02	-1.25E-02	5.00E-03	451.6073387	1806.429355	7225.717419	16257.86419	28902.86967	119.9965884	479.9863534	1919.945414	4319.877181	7679.781654
18	0	-2.50E-02	5.00E-03	0.015632227	0.062528909	0.250115637	0.562760184	1.000462549	0.027591537	0.11036615	0.441464598	0.993295347	1.765858394
19	2.50E-02	-2.50E-02	5.00E-03	477.969502	1911.878008	7647.512032	17206.90207	30590.04813	153.2903108	613.1612433	2452.644973	5518.45119	9810.579693
20	5.00E-02	-2.50E-02	5.00E-03	451.8037029	1807.214811	7228.859246	16264.9333	28915.43698	122.6034548	490.4138192	1961.655277	4413.724373	7846.621107

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

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.

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

Automating 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)	C
2	Copper	400	3.14E-01	
3	Titanium	10	1.79E-01	
4	Aluminum	150	9.75E-02	

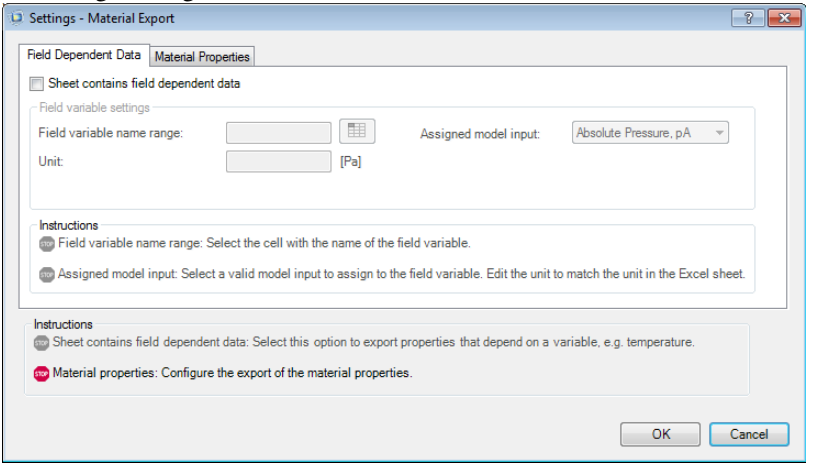
1 Start Excel and open the file `busbar_llexcel_data.xlsx` that you can find in your COMSOL Multiphysics installation directory and in the folder `applications/LiveLink_for_Excel/Tutorials`.

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 5.3 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.

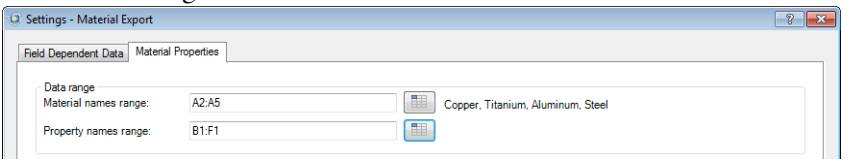
4 In the Field Dependent Data page, make sure that the Sheet contains field dependent data check box is not selected and go to the Material Properties page.

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.

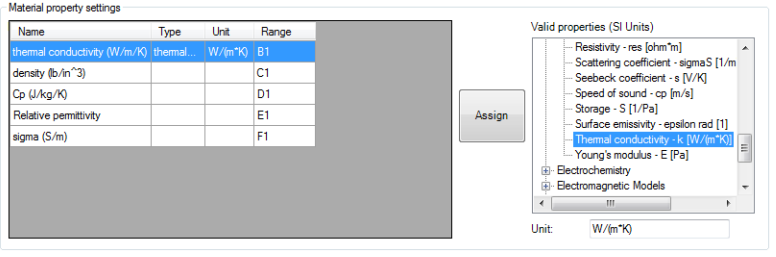
8 Select the range B1:F1 and click OK.



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.

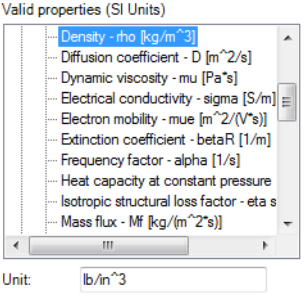


Note that the number of properties remaining to be assigned is displayed in the Instructions section.

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`.

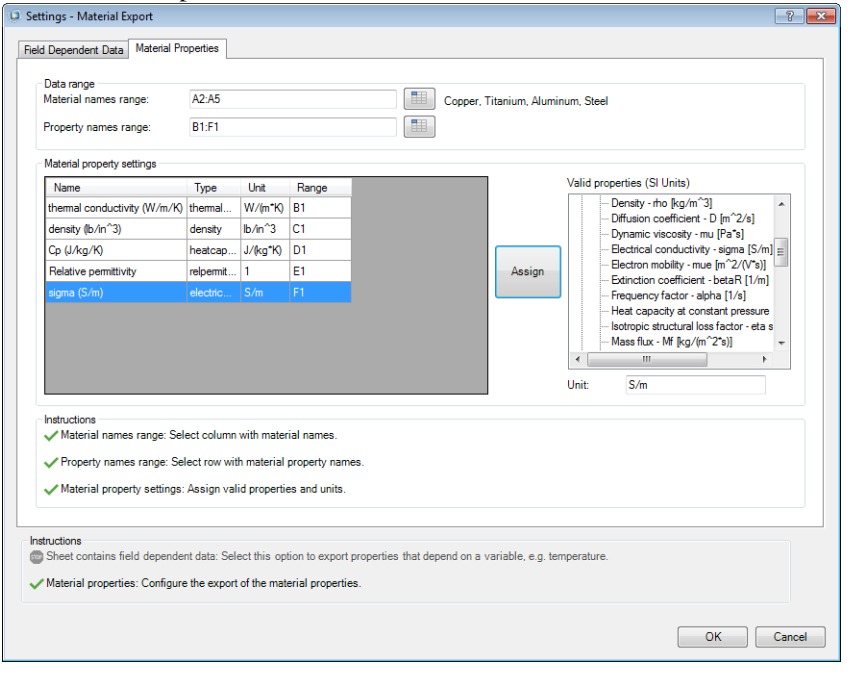


13 Click Assign to assign the properties and the unit to the data in the spreadsheet.

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
7	293.15	0.0010093

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.

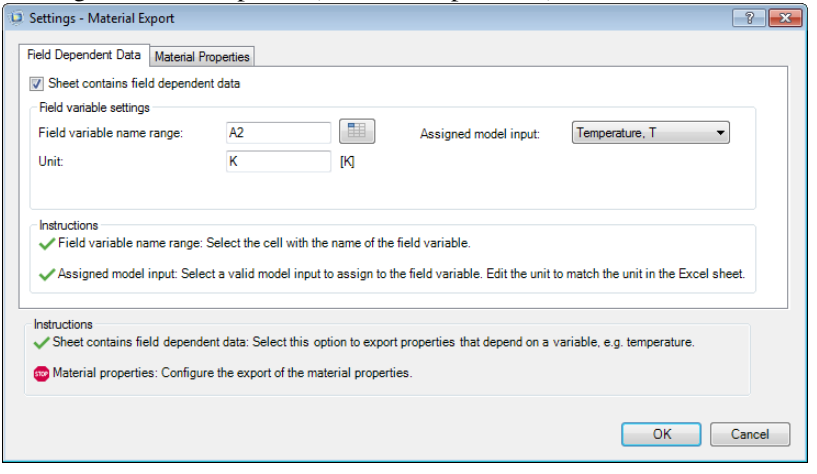
	A	B	C	D	E
1	Water, liquid				
2	T (K)	eta (Pa*s)	Cp (J/(kg*K))	rho (kg/m^3)	k (W/(m*K))
3	273.15	0.0017915	4216.2779	1003.9209	0.55623
4	278.15	0.0015356	4206.8543	1003.0467	0.56642
5	283.15	0.0013248	4198.8844	1002.0375	0.57614
6	288.15	0.0011516	4192.27	1000.8935	0.58541
7	293.15	0.0010093	4186.9181	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 5.3 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.

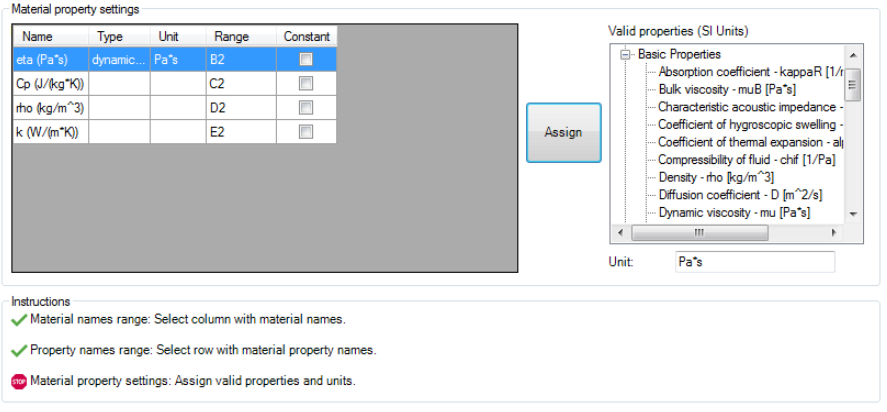
9 Continue by selecting the cells with the property names. Click the Range button ( ) to the right of the Property names range text field.

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.

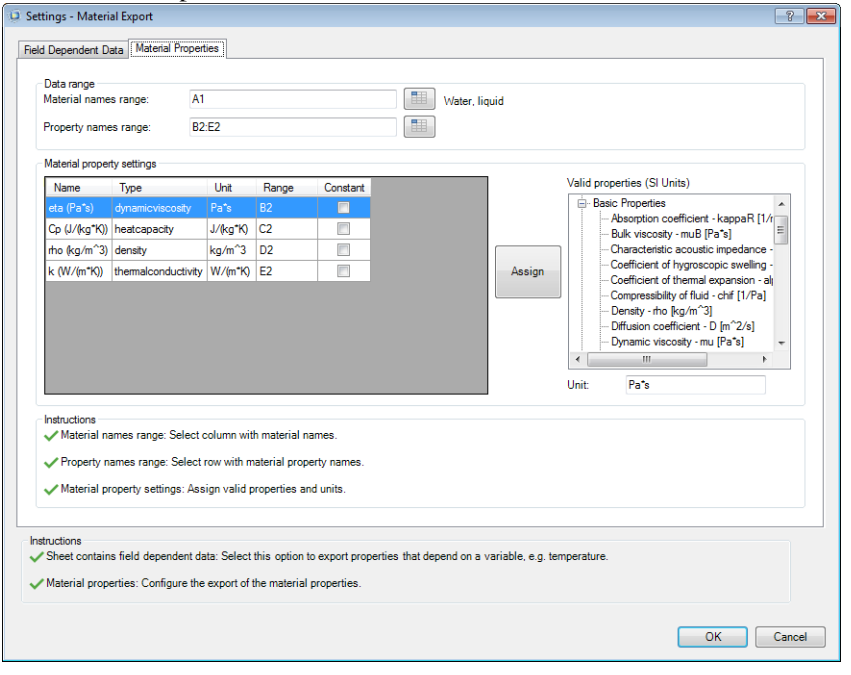


Note that the number of properties remaining to be assigned is displayed in the Instructions section.

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
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

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/v53/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/v53/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 .

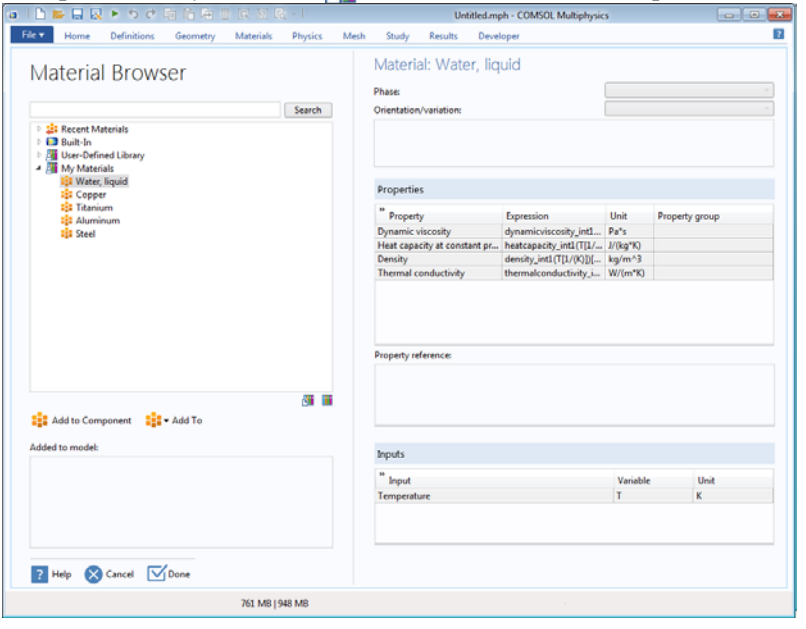
3 For the space dimension click 3D .

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® includes the Excel .XLSX format in the list of supported 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.

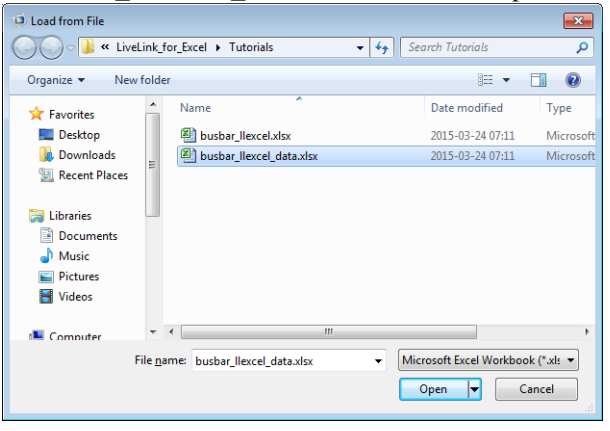
Parameters

Parameters

Name	Expression	Value	Description
L	9[cm]	0.090000 m	
rad_1	6[mm]	0.0060000 m	
tbb	5[mm]	0.0050000 m	
wbb	5[cm]	0.050000 m	
mh	6[mm]	0.0060000 m	
htc	5[W/m^2/K]	5.0000 W/(m²·K)	
Vtot	20[mV]	0.020000 V	

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.

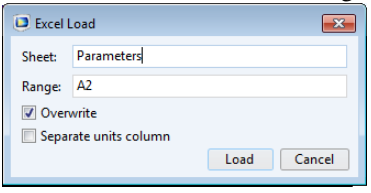
The workbook contains a list of parameters in the Parameters worksheet, which is shown in the figure to the right.

	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.

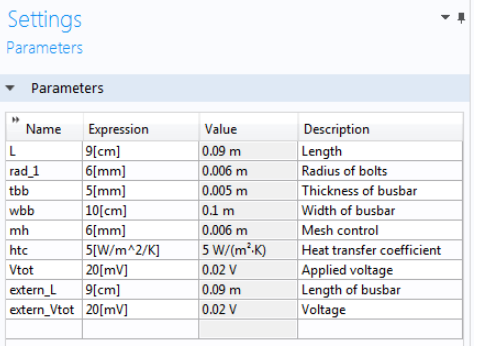


Note: Another way to set which cell to import is to specify the cell range. In the case above it was possible to enter `A2;A5;A7` in the Range text field to import only the parameters `extern_L`, `wbb`, and `extern_Vtot`.

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.



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	0.090000 m	Length
Vtot	extern_Vtot	0.020000 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.

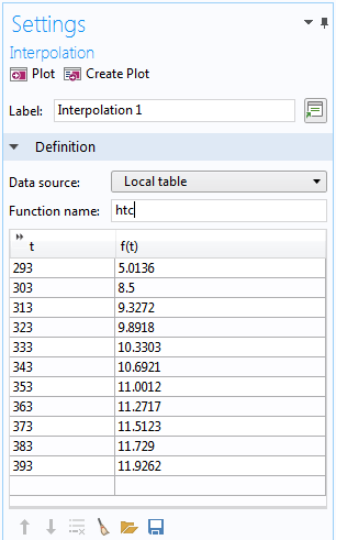
	A	B
1	T[K]	htc[W/m^2/K]
2	293	5.0136
3	303	8.5
4	313	9.3272
5	323	9.8918
6	333	10.3303
7	343	10.6921
8	353	11.0012
9	363	11.2717
10	373	11.5123
11	383	11.729
12	393	11.9262

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.

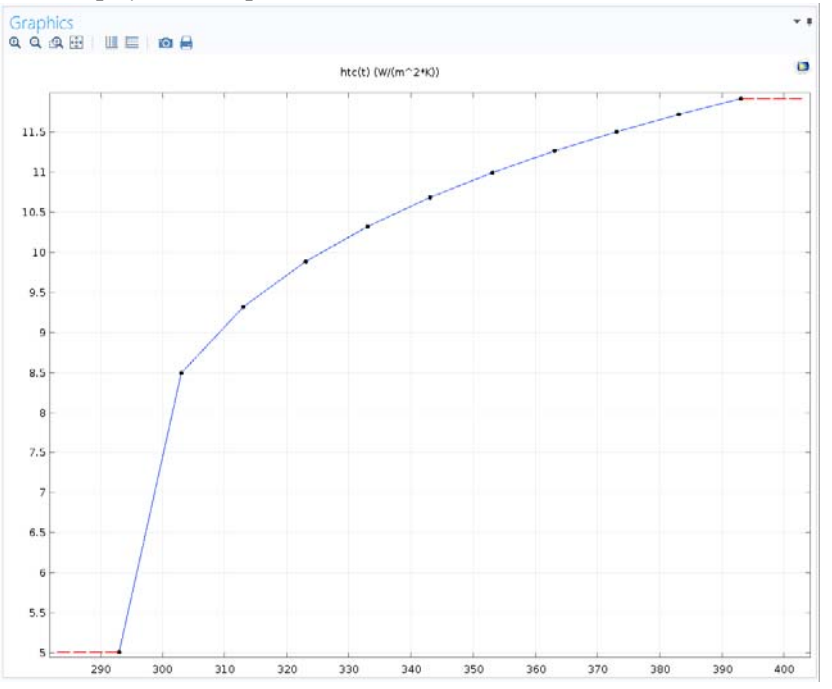
The interpolation table is filled using the data stored in the Excel file.



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.

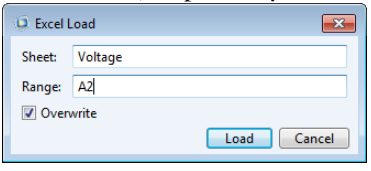
3 Under the Study Extensions section, select Auxiliary sweep and then click the Load from File button.

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
1	Continuation parameter	Parameter value list
2	extern_Vtot	5[mV] 10[mV] 20[mV] 30[mV] 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), click the Temperature (ht) node.