

Introduction to COMSOL Multiphysics®



VERSION 4.3



Introduction to COMSOL Multiphysics

© 1998–2012 COMSOL

Protected by U.S. Patents 7,519,518; 7,596,474; and 7,623,991. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/sla) and may be used or copied only under the terms of the license agreement.

COMSOL, COMSOL Desktop, COMSOL Multiphysics, Capture the Concept, and LiveLink are registered trademarks or trademarks of COMSOL AB. AutoCAD and Inventor are registered trademarks of Autodesk, Inc., in the USA and other countries. LiveLink for AutoCAD and LiveLink for Inventor are not affiliated with, endorsed by, sponsored by, or supported by Autodesk, Inc., and/or its affiliates and/or subsidiaries. MATLAB is a registered trademark of The Mathworks, Inc. Pro/ENGINEER is a registered trademark of Parametric Technology Corporation or its subsidiaries in the U.S. and in other countries. SolidWorks is a registered trademark of Dassault Systèmes SolidWorks Corp. SpaceClaim is a registered trademark of SpaceClaim Corporation. Other product or brand names are trademarks or registered trademarks of their respective holders.

Version:

May 2012

COMSOL 4.3

Contact Information

Visit www.comsol.com/contact for a searchable list of all COMSOL offices and local representatives. From this web page, search the contacts and find a local sales representative, go to other COMSOL websites, request information and pricing, submit technical support queries, subscribe to the monthly eNews email newsletter, and much more.

If you need to contact Technical Support, an online request form is located at www.comsol.com/support/contact.

Other useful links include:

- Technical Support www.comsol.com/support
- Software updates: www.comsol.com/support/updates
- Online community: www.comsol.com/community
- Events, conferences, and training: www.comsol.com/events
- Tutorials: www.comsol.com/products/tutorials
- Knowledge Base: www.comsol.com/support/knowledgebase

Contents

Introduction	5
The COMSOL Desktop.	7
Thorough Example: The Busbar	9
Parameters, Functions, Variables, and Model Couplings.	36
Material Properties and Libraries.	41
Mesh Sequences	43
Adding Physics to a Model.	45
Parameter Sweeps	64
Parallel Computing	71
Geometry Sequences.	73
Easy Example: The Wrench.	85
COMSOL Multiphysics—Additional Concepts.	98
Working with 3D Geometry in the Graphics Window.	105
Keyboard Shortcuts	107
Finding More Information.	109
Supported External File Formats.	111

Introduction

Computer simulation has become an essential part of science and engineering. Digital analysis of components, in particular, is important when developing new products or optimizing designs. Today a broad spectrum of options for simulation is available; researchers use everything from basic programming languages to various high-level packages implementing advanced methods. Though each of these techniques has its own unique attributes, they all share a common concern: Can you rely on the results?

When considering what makes software reliable, it's helpful to remember the goal: you want a model that accurately depicts what happens in the real world. A computer simulation environment is simply a translation of real-world physical laws into their virtual form. How much simplification takes place in the translation process helps to determine the accuracy of the resulting model.

It would be ideal, then, to have a simulation environment that included the possibility to add any physical effect to your model. That is what COMSOL is all about. It's a flexible platform that allows even novice users to model all relevant physical aspects of their designs. Advanced users can go deeper and use their knowledge to develop customized solutions, applicable to their unique circumstances. With this kind of all-inclusive modeling environment, COMSOL gives you the confidence to build the model you want with real-world precision.

Certain characteristics of COMSOL become apparent with use. Compatibility stands out among these. COMSOL requires that every type of simulation included in the package has the ability to be combined with any other. This strict requirement actually mirrors what happens in the real world. For instance in nature, electricity is always accompanied by some thermal effect; the two are fully compatible. Enforcing compatibility guarantees consistent multiphysics models, and the knowledge that, even as the COMSOL family of products expands, you never have to worry about creating a disconnected model again.

Another noticeable trait of the COMSOL platform is adaptability. As your modeling needs change, so does the software. If you find yourself in need of including another physical effect, you can just add it. If one of the inputs to your model requires a formula, you can just enter it. Using tools like parameterized geometry, interactive meshing, and custom solver sequences, you can quickly adapt to the ebbs and flows of your requirements.

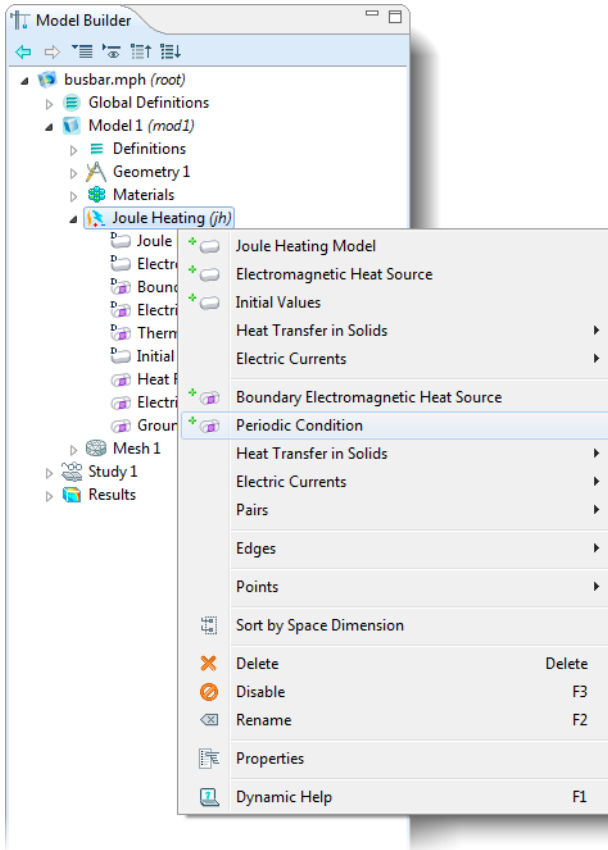
COMSOL Multiphysics also has several problem-solving benefits. When starting a new project, using COMSOL helps you understand your problem. You are able to test out various geometrical and physical characteristics of your model, so you can really hone in on the important design challenges. The flexible nature of the COMSOL environment facilitates further analysis by making “what-if” cases easy to set up and run. You can take your simulation to the production level by optimizing any aspect of your model. Parameter sweeps and target functions can be executed right in the user interface. From start to finish, COMSOL is a complete problem-solving tool.

As you become a more experienced user of COMSOL, your confidence in computer simulation will grow. You will become a more efficient modeler, and the results will show it. The remainder of this introduction is dedicated to give you a strong start toward this goal. After a general introduction to the user interface, several tutorials will take you step by step through sample models that highlight important features. The informative charts give you an idea of COMSOL's capability by associated files, functions, and built-in options. By the end you will be well on your way to reaping all the benefits that COMSOL has to offer.

The COMSOL Desktop

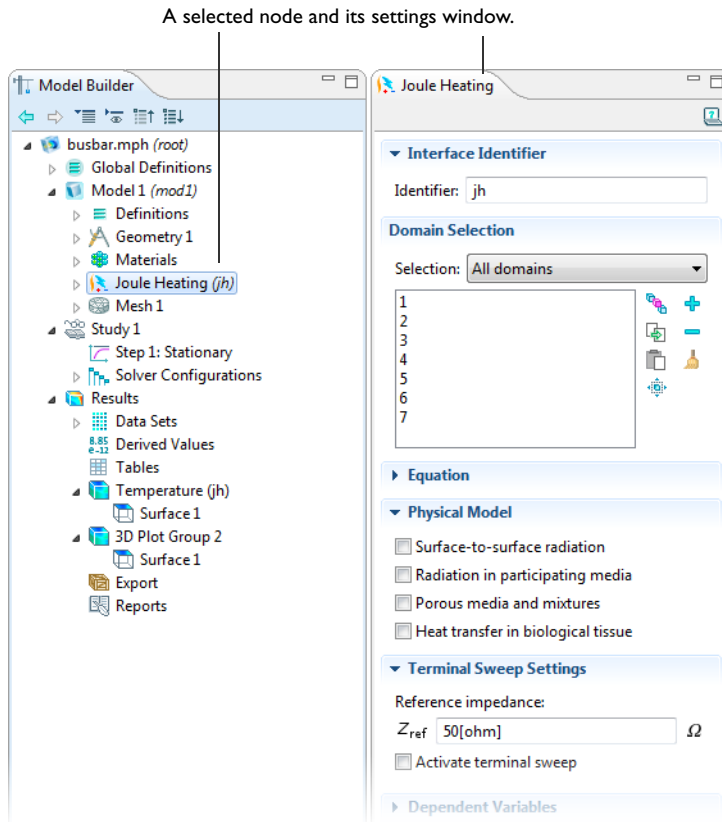
The user interface streamlines the modeling workflow with the Model Builder. Containing a Model Tree, the Model Builder outlines the structure of your model.

Look at this example of the Model Builder. Every step of the modeling process, from defining global variables to the final report of results, is displayed in the Model Tree.



When you right-click any node in the tree, a context menu shows the available features. In addition, many nodes can be easily moved up and down in the model tree by dragging and dropping the nodes. You can also right-click to copy, delete, disable, or enable nodes.

If you choose an action that requires specification, the matching settings window displays next to the Model Builder:



Click any associated node to return to a specific settings window.

As you create the model, each step is shown in the Model Builder. If, for example, your model required a certain sequence of steps to get the right geometry, these are all listed in the order you set. This series of steps can be edited and rerun without having to repeat the entire simulation. Complicated solver sequences you may need for different studies also benefit greatly from this feature.

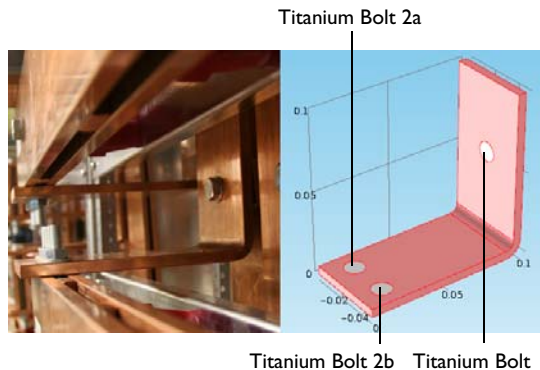
As you work with the COMSOL Desktop and the Model Builder, you will grow to appreciate the organized and streamlined approach. But any description of a user interface is inadequate until you try it for yourself. So in the next few sections, you are invited to work through some examples to familiarize yourself with the software.

Thorough Example: The Busbar

Electrical Heating in a Busbar

In order to get acquainted with COMSOL Multiphysics, it is best to work through a basic example step by step. These instructions describe the essential components of the model building procedure, highlighting several features and demonstrating the common simulation tasks. At the end, you will have built a truly multiphysics model.

The model that you are about to create analyzes a busbar designed to conduct direct current to an electric device (see below). The current conducted in the busbar, from bolt 1 to bolts 2a and 2b, produces heat due to the resistive losses, a phenomenon referred to as Joule heating. The busbar is made of copper while the bolts are made of a titanium alloy. The choice of materials is important because titanium has a lower electrical conductivity than copper and will be subjected to a higher current density.



The goal of your simulation is to precisely calculate how much the busbar heats up. Once you have captured the basic multiphysics phenomena, you will have the chance to investigate thermal expansion yielding structural stresses and strains in the busbar and the effects of cooling by an air stream.

The Joule heating effect is described by conservation laws for electric current and energy. Once solved for, the two conservation laws give the temperature and electric field, respectively. All surfaces, except the bolt contact surfaces, are cooled by natural convection in the air surrounding the busbar. You can assume that the bolt cross-section boundaries do not contribute to cooling or heating of the device. The electric potential at the upper-right vertical bolt surface is 20 mV, and that the potential at the two horizontal surfaces of the lower bolts is 0 V.

Busbar Model Overview


More in depth and advanced topics included with this tutorial are used to show you some of the many options available in COMSOL. The following topics are covered:

- “Parameters, Functions, Variables, and Model Couplings” on page 36, where you learn how to define functions and model couplings.
- “Material Properties and Libraries” on page 41 shows you how to customize a material and add it to your own material library.
- “Mesh Sequences” on page 43 gives you the opportunity to add and define two different mesh nodes and compare two meshes generated and displayed in the Graphics window.
- “Adding Physics to a Model” on page 45 explores the multiphysics capabilities by adding a Solid Mechanics interface and a Laminar Flow interface to the busbar model.
- “Parameter Sweeps” on page 64 gives you some practice of how to make minor changes to the busbar design and quickly view the differences in the Graphics window. Use this analysis to make design changes to a model.
- In the section “Parallel Computing” on page 71, you experiment with adding a Cluster Computing node.
- The last section, “Geometry Sequences” on page 73, contains the step-by-step instructions to build the busbar geometry yourself instead of importing or opening it from a file.

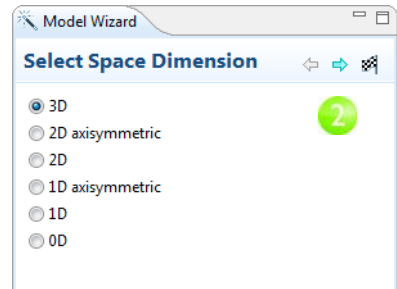
MODEL WIZARD

Open the **Model Wizard**.

To open the Model Wizard, double-click the COMSOL icon on the desktop. Or when COMSOL is already open:

- Click the **New** button  on the main toolbar
- Select **File>New** from the main menu
- Right-click the root node and select **Add Model**

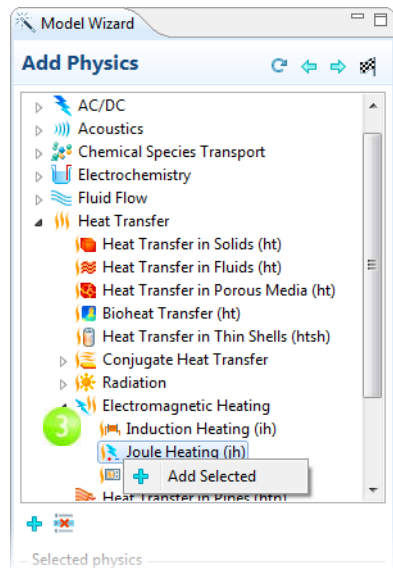
- 2 When the Model Wizard opens, select a space dimension; the default is 3D. Click the **Next** button ➡.



- 3 In the **Add Physics** window, click the **Heat Transfer>Electromagnetic Heating** folder; then right-click **Joule Heating** and choose **Add Selected**. Click the **Next** button ➡.

You can also double-click or click the **Add Selected** button + to add physics.

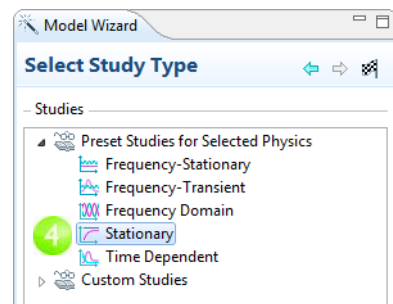
Another way to open the **Add Physics** window is to right-click the **Model** node and select **Add Physics** .



- 4 In the **Select Study Type** window, click to select the **Stationary** study type.


Click the **Finish** button .

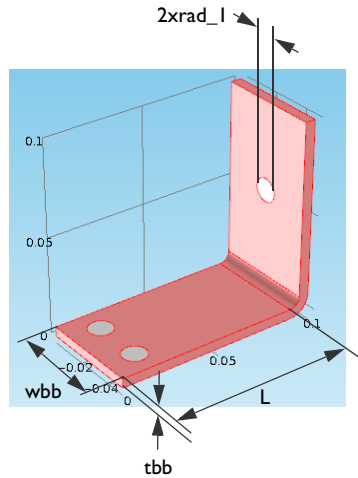
Preset Studies are studies that have solver and equation settings adapted to the selected physics, in this example, Joule heating. Any selection from the **Custom Studies** branch needs manual fine-tuning.



GLOBAL DEFINITIONS


If you want to draw the geometry yourself, the Global Definitions branch is where you define the parameters. First complete the section “Geometry Sequences” on page 73 and then return to this section and use this **busbar.mph** file. If you would prefer to load the geometry from a file, you can just browse through this section and then skip to “Geometry” on page 13.

The Global Definitions node  in the Model Builder stores Parameters, Variables, and Functions with a global scope. You can use these operations in several models. In this case, there is only one Model 1 node where the parameters are used.



Since you will run a geometric parameter study later in this example, define the geometry using parameters from the start. In this step, enter parameters for the length for the lower part of the busbar, **L**, the radius of the titanium bolts, **rad_1**, the thickness of the busbar, **tbb**, and the width of the device, **wbb**.

You will also add the parameters that control the mesh, **mh**, a heat transfer coefficient for cooling by natural convection, **htc**, and a value for the voltage across the busbar, **Vtot**.


- 1 Under **Global Definitions**  click the **Parameters** node **P1**. In the **Parameters** table, click the first row under **Name** and enter **L**.
- 2 Click the first row under **Expression** and enter the value of **L**, **9 [cm]**. You can enter any unit inside the square brackets.

- 3 Continue adding the other parameters: **L**, **rad_1**, **tbb**, **wbb**, **mh**, **htc**, and **Vtot** according to the **Parameters** list. It is a good idea to enter descriptions for variables in case you want to share the model with others and for future reference.

Parameters

Name	Expression	Value	Description
L	9[cm]	0.090000 m	length of busbar
rad_1	6[mm]	0.0060000 m	radius of bolts
tbb	5[mm]	0.0050000 m	thickness of busbar
wbb	5[cm]	0.050000 m	width of busbar
mh	6[mm]	0.0060000 m	mesh control
htc	5[W/m^2/K]	5.0000 W/(m^2·K)	heat transfer coefficient
Vtot	20[mV]	0.020000 V	voltage

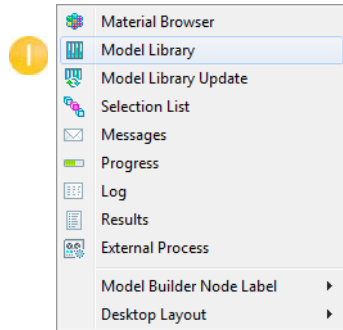
GEOMETRY

In the previous section, “Global Definitions”, you learned how to enter parameters in preparation for drawing the busbar geometry or importing a geometry from another program. The next step is to either open this predefined geometry file from the Model Library or build the busbar geometry yourself. If you want to learn how to use the COMSOL geometry tools and draw the busbar, then click the **Save** button  and name the model **busbar.mph**. Then go to “Geometry Sequences” on page 73.

Alternatively, and if you want to save time, the model geometry can be opened from the Model Library, as described here.

Once a geometry is created, imported, or opened from the Model Library, you can then experiment with different dimensions and update the values of **L**, **tbb**, or **wbb**, and rerun the geometry sequence. The physics, study, parameters, and geometry are included with the model file you are about to open.

- 1 Select **Model Library**  from the main **View** menu.

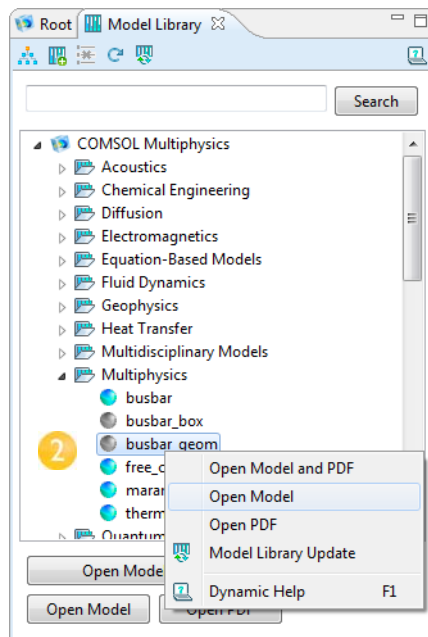


- 2 In the **Model Library** tree under **COMSOL Multiphysics>Multiphysics**, select **busbar_geom**.

To open the model file you can:

- Double-click it
- Right-click and select an option from the menu
- Click one of the buttons under the tree

Now experiment with different geometry expressions for the width parameter, **wbb**.





- 3 If you built the geometry yourself, start from this point to experiment with the geometry expressions using the **busbar.mph** file you saved after completing the steps “Global Definitions” on page 12 and “Geometry Sequences” on page 73.

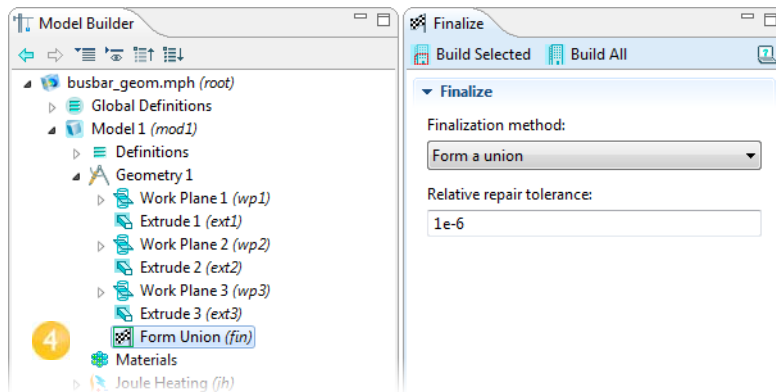
- 3 Under **Global Definitions**  click the **Parameters** node .


In the **Parameters** settings window, click in the **wbb** parameter's **Expression** column and enter **10[cm]** to change the value of the width **wbb**.

3

Parameters		
Name	Expression	Value
L	9[cm]	0.090000 m
rad_1	6[mm]	0.0060000 m
tbb	5[mm]	0.0050000 m
wbb	10[cm]	0.10000 m
mh	6[mm]	0.0060000 m
htc	5[W/m^2/K]	5.0000 W/(m...
Vtot	20[mV]	0.020000 V

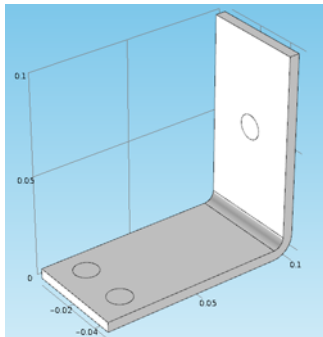
- 4 In the **Model Builder**, click the **Form Union** node  and then the **Build All** button  to rerun the geometry sequence.



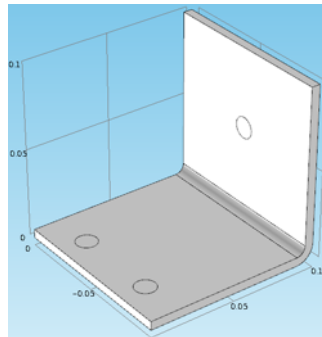
- 5 In the **Graphics** toolbar click the **Zoom Extents** button  to see the wider busbar in the **Graphics** window.



5




wbb=5cm



5

wbb=10cm



6 Experiment with the geometry in the **Graphics** window. Also see “Working with 3D Geometry in the Graphics Window” on page 105.


- To rotate the busbar, left-click and drag it
- To move it, right-click and drag
- To zoom in and out, center-click (and hold) and drag
- To get back to the original position, click the **Go to Default 3D View** button  on the toolbar



6

7 Return to the **Parameters** table and change the value of wbb back to 5 [cm].

8 In the **Model Builder**, click the **Form Union** node  and then the **Build All** button  to rerun the geometry sequence.

9 On the **Graphics** toolbar, click the **Zoom Extents** button .

10 If you built the geometry yourself you are already using the **busbar.mph** file. But if you opened the Model Library file, from the main menu, select **File>Save As** and rename the model **busbar.mph**.

Parameters		
Name	Expression	Value
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/(m...
Vtot	20[mV]	0.020000 V

7




Experienced users of other CAD programs are already familiar with this approach since all major CAD platforms include parameterized geometries. To support this class of users and to avoid redundancy, COMSOL offers the LiveLink™ family of products. These products connect COMSOL Multiphysics directly with a separate CAD program, so that all parameters specified in CAD can be interactively linked with your simulation geometry. The current product line includes LiveLink™ interfaces for SolidWorks®, Inventor®, Pro/ENGINEER®, Creo™ Parametric, AutoCAD®, and SpaceClaim®.

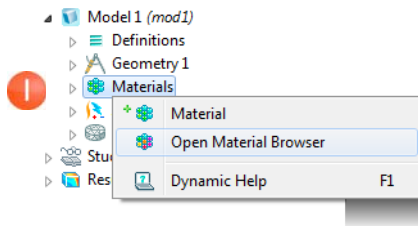
It is also worth noting that the LiveLink™ interface for MATLAB® is available for those who want to incorporate a COMSOL Multiphysics model into an extended programming environment.

After creating or opening the geometry file, it is time to define the materials.

MATERIALS

The Materials node  stores the material properties for all physics and domains in a Model node. The busbar is made of copper and the bolts are made of titanium. Both these materials are available from the Built-In material database.

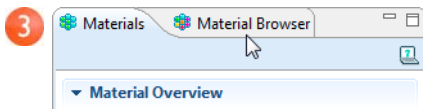
- I In the **Model Builder**, right-click **Materials**  and select **Open Material Browser**  (or select **View>Material Browser**).



- 2 In the **Material Browser**, expand the **Built-In** materials folder and locate **Copper**. Right-click **Copper** and select **+ Add Material to Model**.

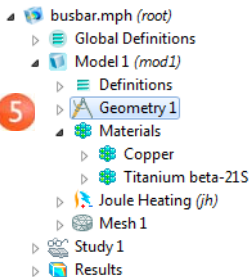
A **Copper** node is added to the **Model Builder**.

- 3 Click the **Material Browser** tab.

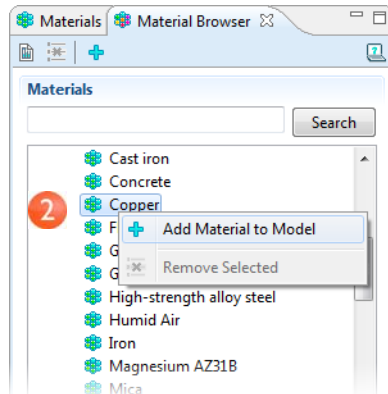
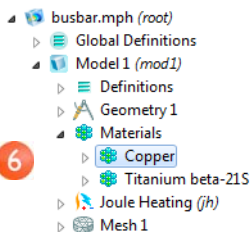


- 4 In the **Material Browser**, scroll to **Titanium beta-21S** in the **Built-In** material folder tree. Right-click and select **+ Add material to model**.

- 5 In the **Model Builder**, collapse the **Geometry 1** node to get an overview of the model.



- 6 Under the **Materials** node, click **Copper**.



7 In the **Material** settings window, examine the **Material Contents** section.

7

Material Contents

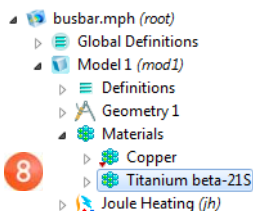
Property	Name	Value	Unit	Property group
✓ Electrical conductivity	sigma	5.998e7[S/m]	S/m	Basic
✓ Heat capacity at constant pr...	Cp	385[J/(kg*K)]	J/(kg...	Basic
✓ Relative permittivity	epsilon	1	1	Basic
✓ Density	rho	8700[kg/m^3]	kg/...	Basic
✓ Thermal conductivity	k	400[W/(m*K)]	W/(...	Basic
Relative permeability	mu	1	1	Basic
! Coefficient of thermal expa...	alpha	17e-6[1/K]	1/K	Basic
Young's modulus	E	110e9[Pa]	Pa	Young's modulus and P...
Poisson's ratio	nu	0.35	1	Young's modulus and P...
Reference resistivity	rho0	1.72e-8[ohm...	Ω*m	Linearized resistivity

The **Material Contents** section has useful feedback about the model's material property usage. Properties that are both required by the physics and available from the material are marked with a green check mark ✓. Properties required by the physics but missing in the material result in an error and are marked with a warning sign ⚠. A property that is available but not used in the model is unmarked.

! The **Coefficient of thermal expansion** property is not used, but it is needed later when heat induced stresses and strains are added to the model (in the section “Adding Solid Mechanics” on page 45).

Because the copper material is added first, by default all parts have copper material assigned. In the next step you will assign titanium properties to the bolts, which overrides the copper material assignment for those parts.

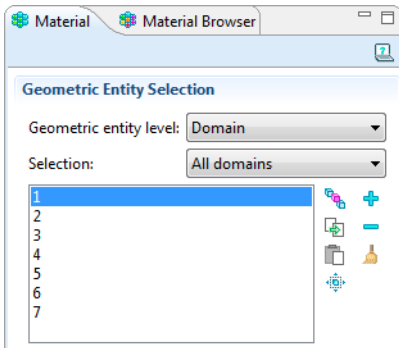
8 In the **Model Builder**, click **Titanium beta-21S**.



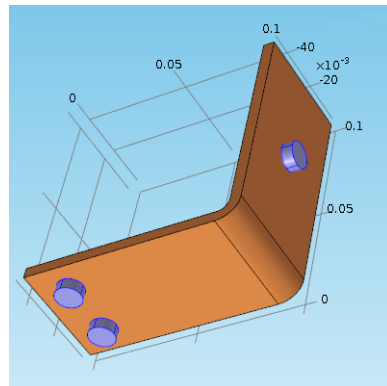
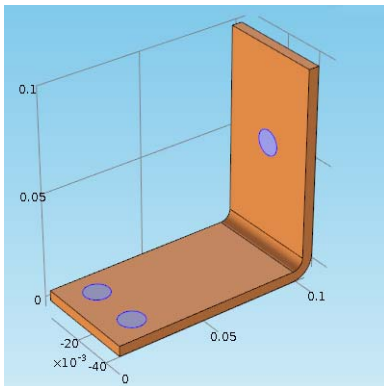
- 9 Select **All Domains** from the **Selection** list and then click domain 1 in the list. Now remove Domain 1.

To remove a domain (or any geometric entity such as boundaries, edges, or points), you can:

- Click domain 1 in the selection list found in the **Material** settings window, then click the **Remove from Selection** button —
- In the **Graphics** window, click domain 1 and right-click to remove it from the selection list

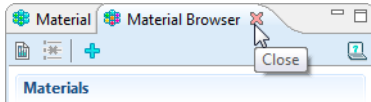


- 9 Cross-check: Domains 2, 3, 4, 5, 6, and 7.

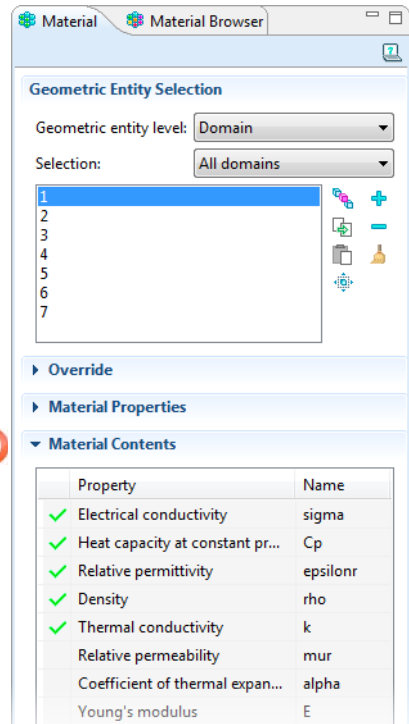


10 Be sure to inspect the **Material Contents** section in the settings window. All the properties used by the physics interfaces should have a green check mark ✓.

Close the **Material Browser**.



10

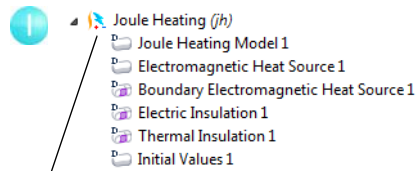


PHYSICS

The domain settings for the Joule Heating interface are complete now that you have set the material properties for the different domains. Next you will set the boundary conditions for the heat transfer problem and the conduction of electric current.

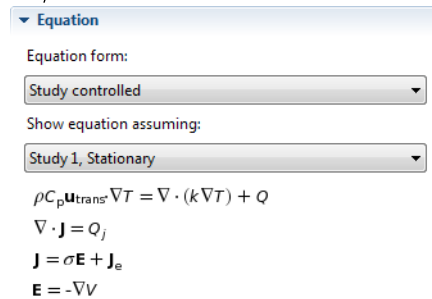
In the **Model Builder**, expand the **Joule Heating** node to examine the default physics nodes.

The 'D' in the upper left corner of a node means it is a default node.



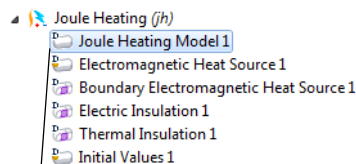
The equations that COMSOL solves are displayed in the **Equation** section of the settings window. The equations change based on the **Equation form** selected.

The default equation form is inherited from the study added in the Model Wizard. For the Joule Heating node, COMSOL displays the equations solved for the temperature and electric potential.

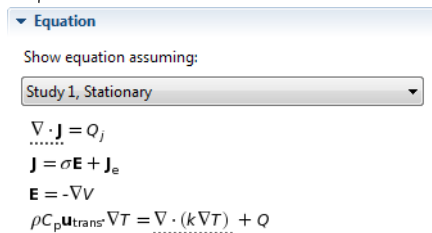


To always display the section in its expanded view, click the **Expand Sections** button (☰) on the **Model Builder** toolbar and select **Equations**. Selecting this option expands all the Equation sections on physics settings windows.



The domain level **Joule Heating Model 1** node has the settings for heat conduction and current conduction.

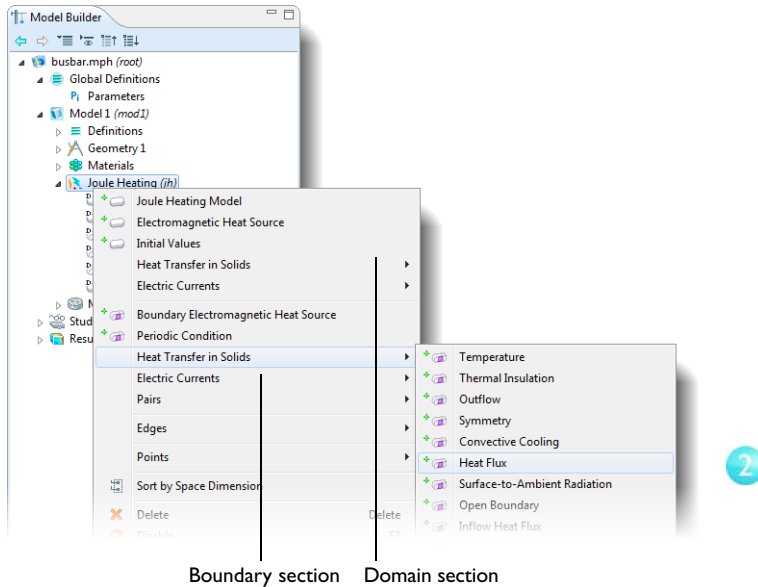


The contribution of the **Joule Heating Model 1** node to the entire equation system is underlined in the **Equation** section.

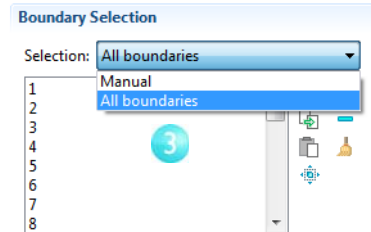


The heating effect for Joule heating is set in the **Electromagnetic Heat Source 1** node. The **Thermal Insulation 1** node contains the default boundary condition for the heat transfer problem and **Electric Insulation 1** corresponds to the conservation of electric current. The **Initial Values 1** node contains initial guesses for the nonlinear solver for stationary problems and initial conditions for time-dependent problems.

- 2 Right-click the **Joule Heating** node . In the second section of the context menu—the boundary  section—select **Heat Transfer in Solids>Heat Flux**.



- 3 In the **Heat Flux** settings window, select **All boundaries** from the **Selection** list.
Assume that the circular bolt boundaries are neither heated nor cooled by the surroundings.



In the next step these boundaries are removed from the heat flux selection list, which leaves them with the default insulating boundary condition for the Heat Transfer interfaces.

- 4 Rotate the busbar to view the back. Click one of the circular titanium bolt surfaces to highlight it in green. Right-click anywhere in the **Graphics** window to remove this

4

5

Heat Flux

☐ General inward heat flux

☒ Inward heat flux

$q_0 = h \cdot (T_{\text{ext}} - T)$



Heat transfer coefficient:

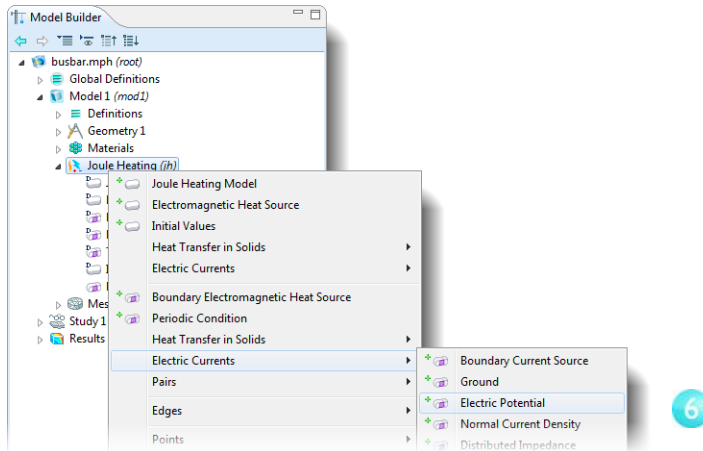
h W/(m² K)

External temperature:

T_{ext} K

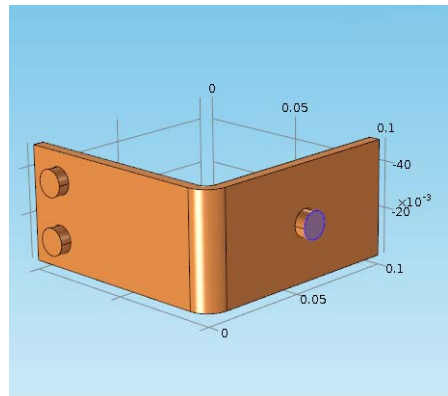
24 | Thorough Example: The Busbar

- 6 In the **Model Builder**, right-click the **Joule Heating** node . In the second section of the context menu—the boundary section—select **Electric Currents>Electric Potential**. An **Electric Potential**  node is added to the **Model Builder**.

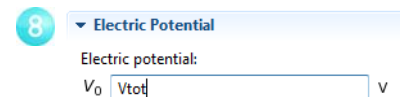


- 7 Click the circular face of the upper titanium bolt to highlight it and right-click anywhere to add it (boundary 43) to the **Selection** list.



- 7 Cross-check: Boundary 43.

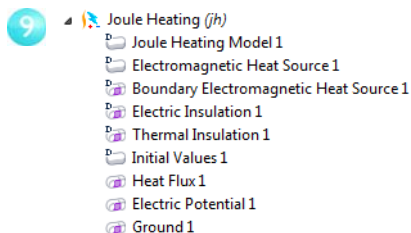


- 8 In the **Electric Potential** settings window, enter V_{tot} in the **Electric potential** field.



The last step is to set the two remaining bolt surfaces to ground.

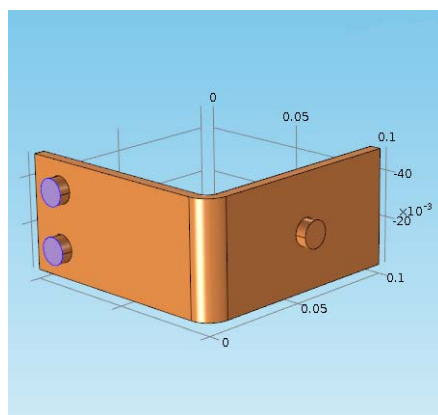
- 9 In the **Model Builder**, right-click the **Joule Heating** node . In the boundary section of the context menu, select **Electric Currents>Ground**. A **Ground** node  is added to the Model Builder. The node sequence under **Joule Heating** should match this figure.



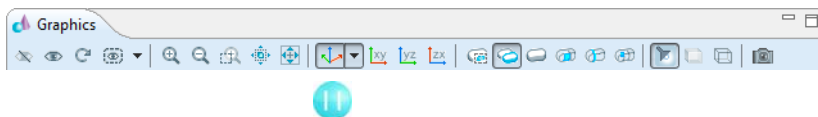
- 10 In the **Graphics** window, click one of the remaining bolts to highlight it. Right-click anywhere to add it to the **Selection** list.

Repeat this step to add the last bolt. Boundaries 8 and 15 are added to the selection list.

- 10 Cross-check: Boundaries 8 and 15.



- 11 On the **Graphics** toolbar, click the **Go to Default 3D View** button .



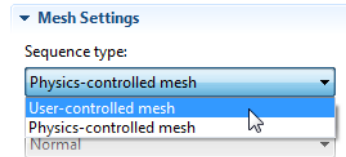
MESH

The simplest way to mesh is to create an unstructured tetrahedral mesh, which is perfect for the busbar. Alternatively, you can create several mesh sequences as shown in “Mesh Sequences” on page 43.



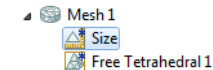
A physics-controlled mesh is created by default. In most cases, it is possible to skip to the Study branch and just solve the model. For this exercise, the settings are investigated in order to parameterize the mesh settings.

- 1 In the **Model Builder**, click the **Mesh 1** node . In the **Mesh** settings window, select **User-controlled mesh** from the **Sequence type** list.



- 2 Under **Mesh 1**, click the **Size** node .

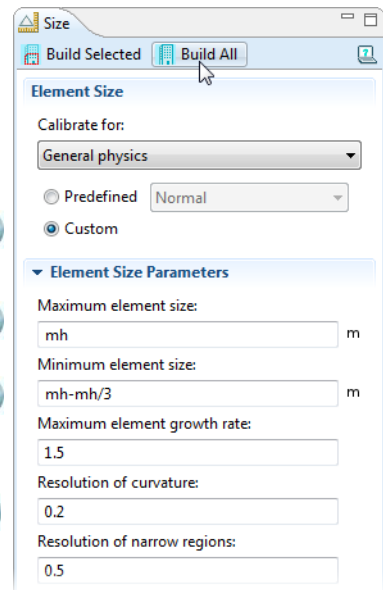
The asterisk (*) that displays in the upper-right corner of the icon indicates that the node is being edited.



- 3 In the **Size** settings window under **Element Size**, click the **Custom** button.

Under **Element Size Parameters**, enter:

- mh in the **Maximum element size** field. Notice that mh is 6 mm—the value entered earlier as a global parameter. By using the parameter mh , the variations in element size are limited.
- $mh - mh/3$ in the **Minimum element size** field. The **Minimum element size** is slightly smaller than the maximum size.
- **0.2** in the **Resolution of curvature** field. The resolution of curvature determines the number of elements on curved edges: the larger this resolution value is, the more elements are used.



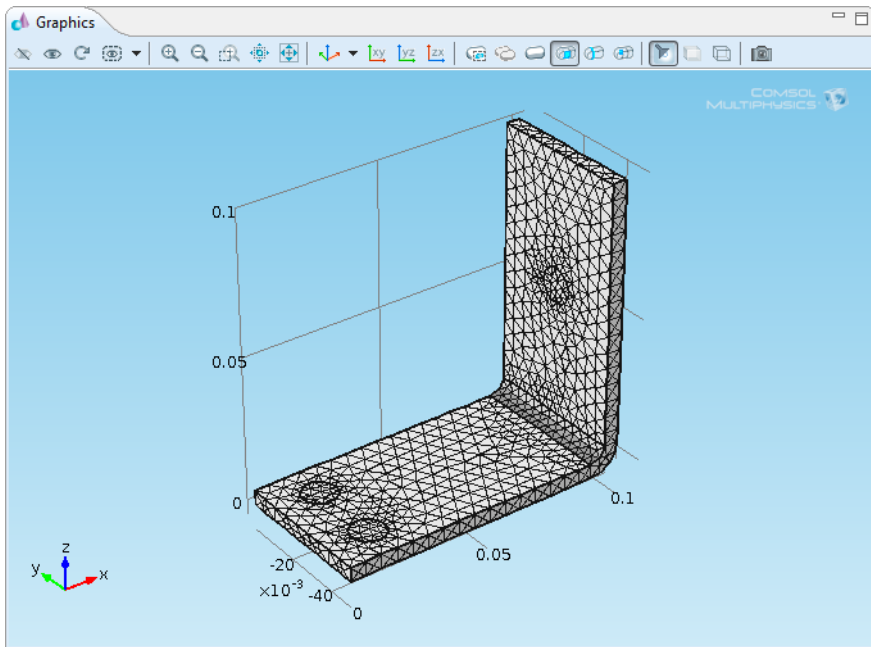
The **Maximum element growth rate**

determines how fast the elements should grow from small to large over a domain. The larger this value is, the larger the growth rate. A value of **1** does not give any growth.



The **Resolution of narrow regions** works in a similar way to the resolution of curvature.


- 4 Click the **Build All** button  in the **Size** settings window to create the mesh as in this figure:

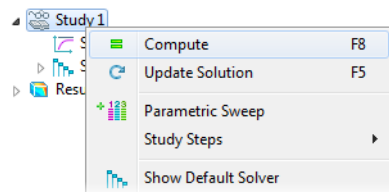
4



STUDY


- I To run a simulation, in the **Model Builder**, right-click **Study 1**  and choose **Compute** . Or press F8.

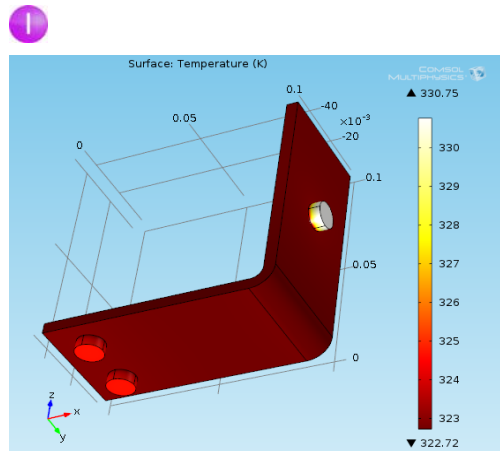
The **Study** node  automatically defines a solution sequence for the simulation based on the selected physics and the study type. The simulation only takes a few seconds to solve.





RESULTS

The default plot displays the temperature in the busbar. The temperature difference in the device is less than 10 K due to the high thermal conductivity of copper and titanium. The temperature variations are largest on the top bolt, which conducts double the amount of current compared to the two lower bolts. The temperature is substantially higher than the ambient temperature of 293 K.

- 1 Click and drag the image in the **Graphics** window to rotate and view the back of the busbar.
- 2 On the **Graphics** toolbar, click the **Go to Default 3D View** button .



You can now manually set the color table range to visualize the temperature difference in the copper part.

- 3 In the **Model Builder**, expand the **Results>Temperature** node  and click the **Surface 1** node .

- 4 In the **Surface** settings window, click **Range** to expand the section. Select the **Manual color range** check box and enter **323** in the **Maximum** field (replace the default).

4

The screenshot shows the 'Surface' settings window with the 'Plot' tab selected. The 'Data' section shows 'Data set: From parent'. The 'Expression' section shows 'Expression: T' and 'Unit: K'. The 'Title' section is empty. The 'Range' section is expanded, and the 'Manual color range' checkbox is checked. The 'Minimum' value is 322.72382 and the 'Maximum' value is 323. The 'Manual data range' checkbox is unchecked.

Surface Plot

Data

Data set: From parent

Expression

Expression: T

Unit: K

Description: Temperature

Title



Range

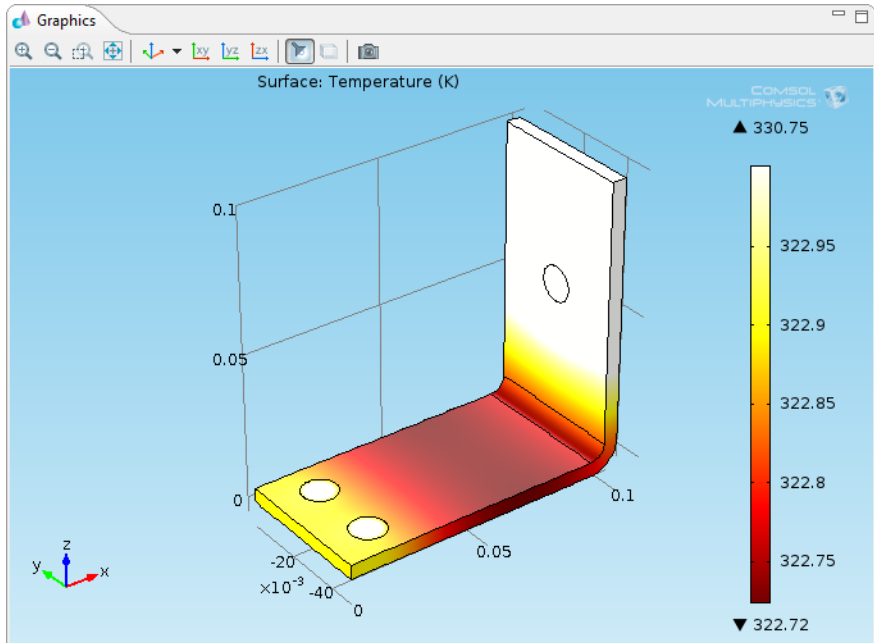
☒ Manual color range

Minimum: 322.72382

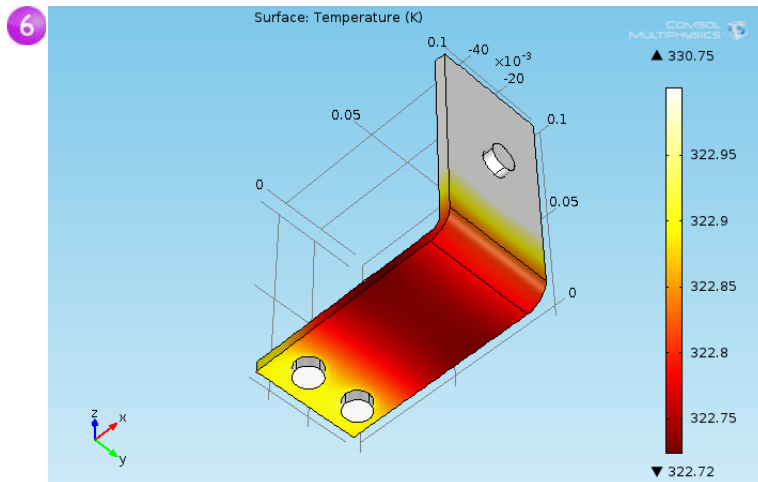
Maximum: 323

☐ Manual data range

- 5 Click the **Plot** button  on the **Surface** settings window. On the **Graphics** toolbar, click the **Zoom Extents** button  to view the updated plot.



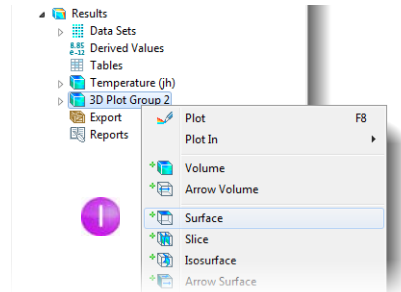
- 6 Click and drag in the **Graphics** window to rotate the busbar and view the back.



The temperature distribution is symmetric with a vertical mirror plane running between the two lower titanium bolts and running across the middle of the upper bolt. In this case, the model does not require much computing power and you can model the whole geometry. For more complex models, you can consider using symmetries in order to reduce the size of the model.

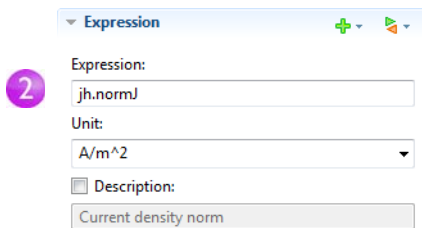
The next Surface plot generated shows the current density in the device.

- 1 In the **Model Builder**, right-click **Results** and add a **3D Plot Group**. Right-click **3D Plot Group 2** and add a **Surface** node.



- 2 In the **Surface** settings window under **Expression**, click the **Replace Expression** button. Select **Joule Heating (Electric Currents)>Currents and charge>Current density norm (jh.normJ)**.

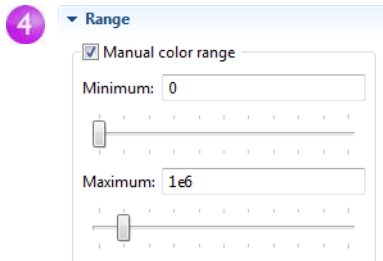
$j_{h-normJ}$ is the variable for the magnitude, or absolute value, of the current density vector. You can also enter $j_{h-normJ}$ in the **Expression** field when you know the variable name.




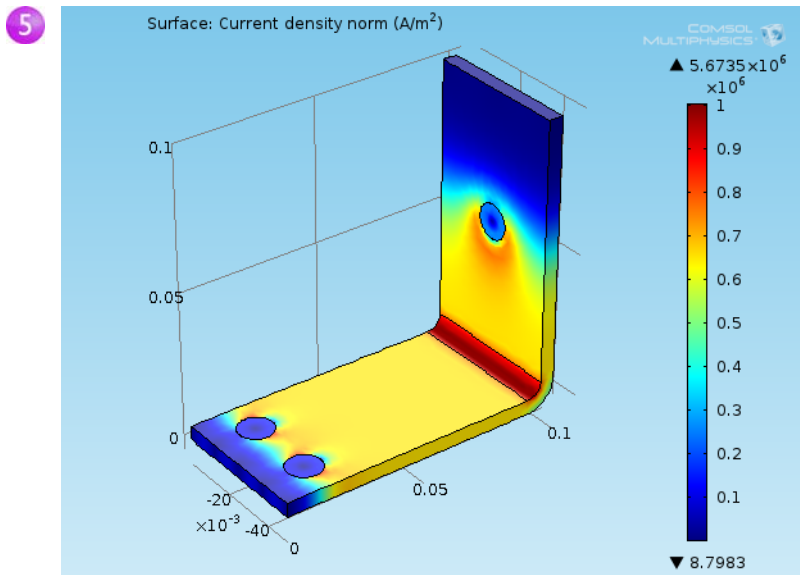
- 3 Click the **Plot** button.

The plot that displays in the **Graphics** window is almost uniform in color due to the high current density at the contact edges with the bolts. The next step is to manually change the color table range to visualize the current density distribution.

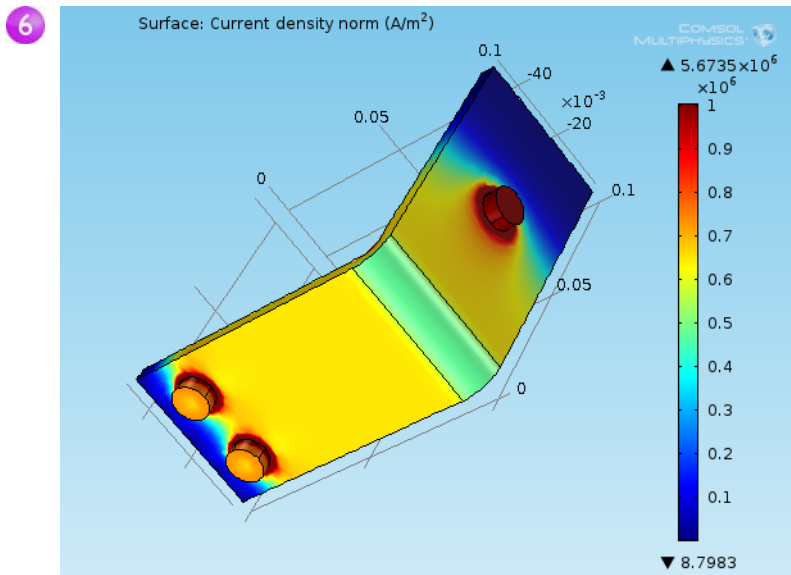
- 4 On the **Surface** settings window under **Range**, select the **Manual color range** check box. Enter **1e6** in the **Maximum** field and replace the default.



- 5 Click the **Plot** button . The plot automatically updates in the **Graphics** window. The resulting plot shows how the current takes the shortest path in the 90-degree bend in the busbar. Notice that the edges of the busbar outside of the bolts are hardly utilized for current conduction.



- 6 Click and drag the busbar in the **Graphics** window to view the back. Continue rotating the image to see the high current density around the contact surfaces of each of the bolts.





- ! Make sure to save the model. This version of the model, **busbar.mph**, is reused and renamed during the next set of tutorials.

When you are done, click the **Go to Default 3D View** button  on the **Graphics** toolbar and create a model thumbnail image.

Creating Model Images from Plots

With any solution, you can create an image to display in COMSOL when browsing for model files. After generating a plot select **File>Save Model Thumbnail** from the main menu.

There are two other ways to create images from a plot. One is to click the **Image Snapshot** button  in the Graphics toolbar to directly create an image. You can also add an *Image* node  to a *Report* by right-clicking the plot group of interest.

This completes the first part of the introduction to a multiphysics simulation. The next sections are designed to increase your understanding of the steps you implemented up to this point as well as to extend your simulation to include other

relevant effects, like thermal expansion and fluid flow. These additional tutorial sections start on the following pages:

- “Parameters, Functions, Variables, and Model Couplings” on page 36
- “Material Properties and Libraries” on page 41
- “Mesh Sequences” on page 43
- “Adding Physics to a Model” on page 45
- “Parameter Sweeps” on page 64
- “Parallel Computing” on page 71
- “Geometry Sequences” on page 73

Parameters, Functions, Variables, and Model Couplings

This section explores working with Parameters, Functions, Variables, and Model Couplings.

Global Definitions and Definitions contain functionality that help you to prepare model inputs and model couplings and to organize simulations. You have already used the functionality for adding Parameters to organize model inputs in “Global Definitions” on page 12.

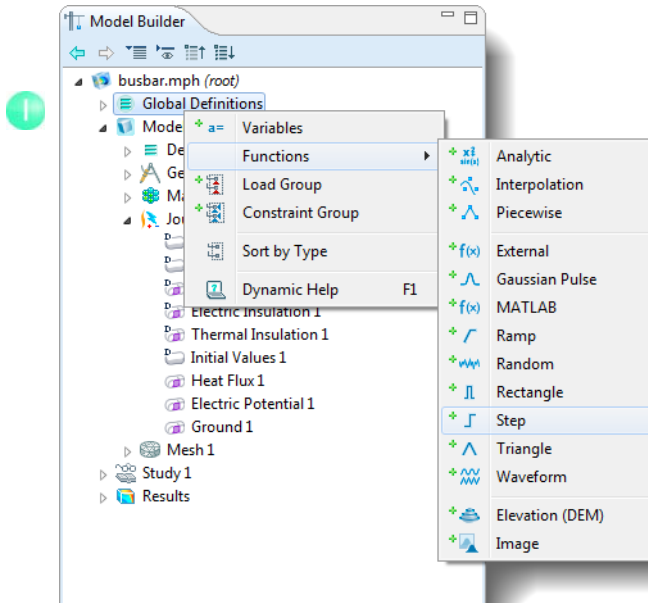
Functions, available as both Global Definitions and Definitions, contain a set of predefined functions that can be useful when setting up multiphysics simulations. For example, the Step function can create a smooth step function for defining different types of switches.

To illustrate using functions, assume that you want to add a time dependent study to the busbar model by applying an electric potential across the busbar that goes from 0 V to 20 mV in 0.5 seconds. For this purpose, you could use a step function to be multiplied with the parameter V_{tot} . Add a function that goes smoothly from 0 to 1 in 0.5 seconds to find out how functions can be defined and verified.

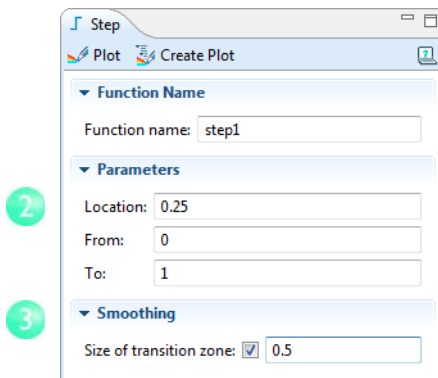
DEFINING FUNCTIONS

For this section, you can continue working with the same model file created in the previous section. Locate and open the file **busbar.mph** if it is not already open on the COMSOL Desktop.

- 1 Right-click the **Global Definitions** node and select **Functions>Step**.

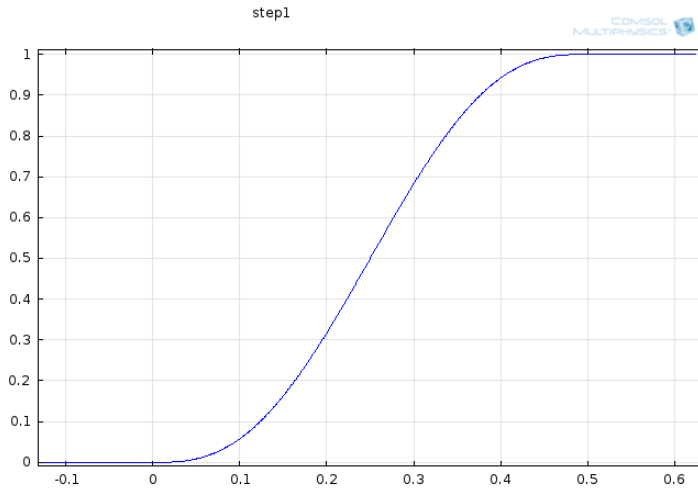


- 2 In the **Step** settings window, enter **0.25** in the **Location** field to set the location of the middle of the step, where it has the value of **0.5**.





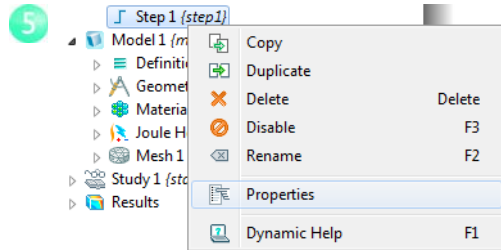
- 3 Click **Smoothing** to expand the section and enter **0.5** in **Size of the Transition zone** to set the width of the smoothing interval.
- 4 Click the **Plot** button in the **Step** settings window.

If your plot matches the one below, this confirms that you have defined the function correctly.

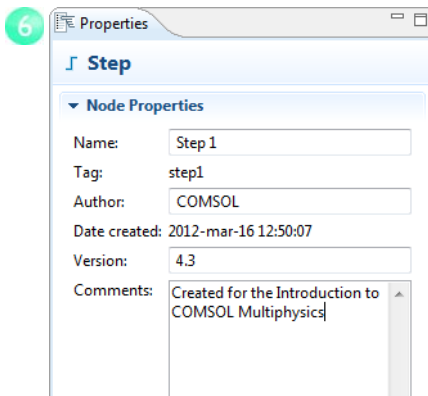





You can also add comments and rename the function to make it more descriptive.

- 5 Right-click the **Step 1** node  in the **Model Builder** and select **Properties** .



- 6 In the **Properties** window, enter any information you want.



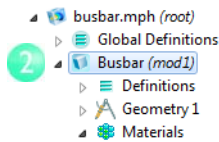
The **Global Definitions**  and **Definitions**  nodes can contain **Variables** , which are expressions of the dependent variables—the variables that are solved for in a simulation. You can define global variables that can be used in several models.

For the purpose of this exercise, assume that you want to introduce a second model to represent an electric device connected to the busbar through the titanium bolts.

A first step would be to rename Model 1 to specify that it represents the busbar:


1 Right-click the **Model 1** node  and select **Rename**  (or press F2).

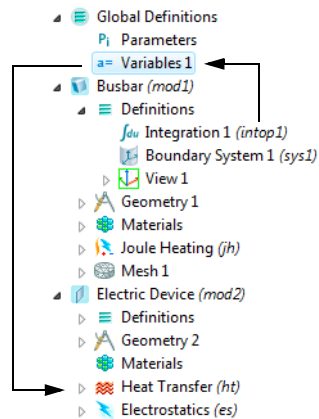
2 In the **Rename Model** window, enter **Busbar**. Click **OK** and save the model.







DEFINING MODEL COUPLINGS

The next steps are for information only and you do not need to reproduce them unless you want to.

Click the **Definitions** node  under **Busbar (mod1)** to introduce a Model Coupling that integrates any **Busbar (mod1)** variable at the bolt boundaries facing the electric device. You can use this coupling to define a Variable—in **Global Definitions**—that calculates the total current. This variable is then globally accessible and could, for example, form a boundary condition for the current that is fed to the electric device in the **Electric Device (mod2)** node.






The Model Couplings in Definitions have a wide range of use. The **Average** , **Maximum** , and **Minimum**  model couplings have applications in generating results as well as in boundary conditions, sources, sinks, properties, or any other contribution to the model equations. The **Probes**  are for monitoring the solution

progress. For instance, you can follow the solution in a critical point during a time-dependent simulation or at parameter value in a parametric study.

You can also use Model Couplings to map variables from one face in a model to another (extrusion couplings) or to integrate a variable along curves and map from one entity to another (projection couplings).

You can find an example of using the average operator in “Parameter Sweeps” on page 64. Also see “Built-in Functions” on page 114, for a list of available COMSOL functions.



To learn more about working with definitions, in the **Model Builder** click the **Definitions**  or **Global Definitions**  node and press F1 to open the **Dynamic Help** window . This window displays help about the selected item in the COMSOL Desktop and provides links to the documentation. It could take up to a minute for the window to load the first time it is activated but the next time it will load quickly.

Material Properties and Libraries

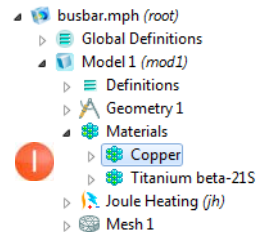
Up to now, you have used the functionality in Materials to access the properties of copper and titanium in the busbar model. In Materials, you are also able to define your own materials and save them in your own material library. You can also add material properties to existing materials. In cases where you define properties that are functions of other variables, typically temperature, the plot functionality helps you to verify the property functions in the range of interest.

First investigate how to add a property to an existing material. Assume that you want to add bulk modulus and shear modulus to the copper properties.

CUSTOMIZING MATERIALS

Locate and open the file `busbar.mph` if it is not already open on the COMSOL Desktop.

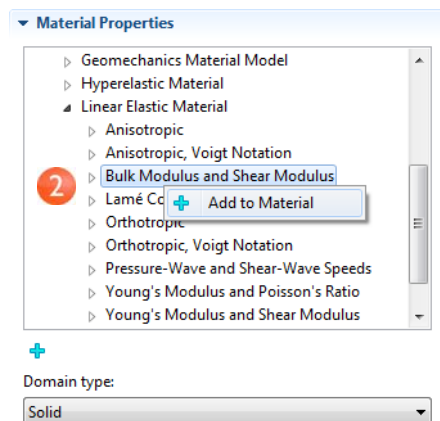
- In the **Model Builder**, under **Materials**, click **Copper**.



- In the **Material** settings window, the **Materials Properties** section contains a list of all the definable properties.

Expand the **Solid Mechanics>Linear Elastic Material** section. Right-click **Bulk Modulus and Shear Modulus** and select **Add to Material**.

This lets you define the bulk modulus and shear modulus for the copper in your model.



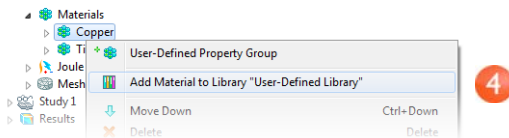
- 3 Locate the **Material Contents** section. **Bulk modulus** and **Shear modulus** rows are now available in the table. The warning sign ⚠ indicates the values are not yet defined. To define the values, click the **Value** column. In the **Bulk modulus** row, enter **140e9** and in the **Shear modulus** row, enter **46e9**.

3 ▼ **Material Contents**

Property	Name	Value
⚠ Bulk modulus	K	
⚠ Shear modulus	G	
Relative permeability	mur	1
Electrical conductivity	sigma	5.998e7[S/m]
Coefficient of thermal expansion	alpha	17e-6[1/K]
Heat capacity at constant pressure	Cp	385[J/(kg*K)]
Relative permittivity	epsilon _r	1
Density	rho	8700[kg/m^3]
Thermal conductivity	k	400[W/(m*K)]

By adding these material properties, you have changed the Copper material. However, you cannot save this in the read-only **Solid Mechanics** material library. However, you can save it to your own material library.

- 4 In the **Model Builder**, right-click **Copper** and select **Add Material to “User Defined Library”**.



- 5 Right-click **Materials** and select **Open Material Browser**. In the **Material Browser**, right-click **User Defined Library** and select **Rename Selected**.
- 6 Enter **My Materials** in the **Enter New Name** dialog box.

Mesh Sequences

A model can contain different mesh sequences to generate meshes with different settings. These sequences can then be accessed by the study steps. In the study, you can select which mesh you would like to use in a particular simulation.

In the busbar model, a second mesh node is added to create a mesh that is refined in the areas around the bolts and the bend. You do this by increasing the mesh resolution on curved surfaces.

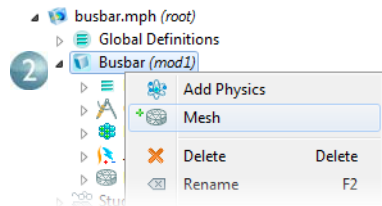
ADDING A MESH

Locate and open the file **busbar.mph** (if it is not already open on the COMSOL Desktop).

1 In order to keep this model in a separate file for later use, from the main menu, select **File>Save as** and rename the model **busbar_I.mph**.

2 To add a second mesh node, right-click the **Busbar (mod1)** node and select **Mesh**.

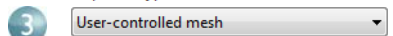
By adding another **Mesh** node, it creates a **Meshes** parent node that contains both **Mesh 1** and **Mesh 2**.



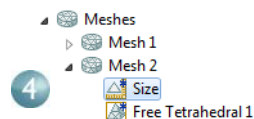
! The **Model 1** node was renamed to **Busbar** in a previous step. To display different combinations of node names, tags, and identifiers, select **View>Model Builder Node Label** and select an option from the list.

3 Click the **Mesh 2** node. In the **Mesh** settings window under **Mesh Settings**, select **User-controlled mesh** as the **Sequence type**.


A **Size** and **Free Tetrahedral** node are added under **Mesh 2**.



4 In the **Model Builder**, under **Mesh 2**, click **Size**.



The asterisks in the upper-right corner indicate that the nodes are being edited.

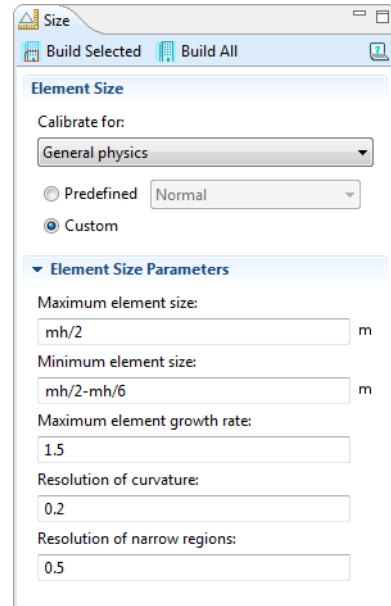
- 5 In the **Size** settings window under **Element Size**, click the **Custom** button.
- 6 Under **Element Size Parameters**, enter:
 - $mh/2$ in the **Maximum element size** field
 - $mh/2-mh/6$ in the **Minimum element size** field
 - 0.2 in the **Resolution of curvature** field
- 7 Click the **Build All** button .

5

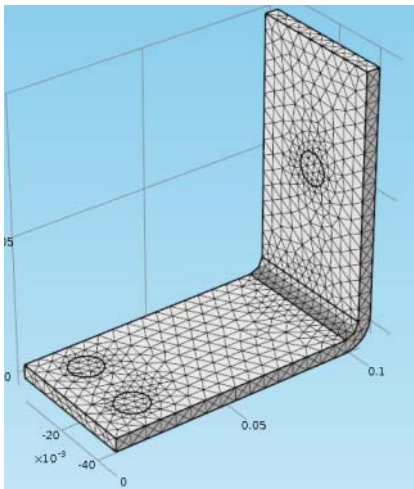
6

6

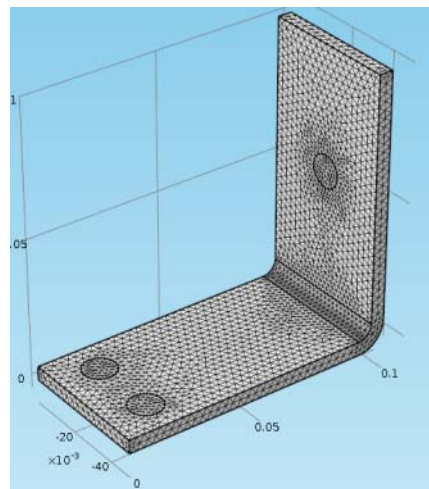
6



Compare Mesh 1 and Mesh 2 by clicking the **Mesh** nodes. The mesh is updated in the Graphics window. An alternative for using many different meshes is to run a parametric sweep of the parameter for the maximum mesh size, mh , that was defined in the section “Global Definitions” on page 12.



Mesh 1



Mesh 2

Adding Physics to a Model

COMSOL's distinguishing characteristics of adaptability and compatibility are prominently displayed when you add physics to an existing model. In this section, you will understand the ease with which this seemingly difficult task is performed. By following these directions, you can add structural mechanics and fluid flow to the busbar model.

STRUCTURAL MECHANICS

After completing the busbar Joule heating simulation, it is known that there is a temperature rise in the busbar. The next logical question to ask is: What kind of mechanical stress is induced by thermal expansion? To answer this question, COMSOL Multiphysics makes it easy for you to expand your model to include the physics associated with structural mechanics.





To complete these steps, either the Structural Mechanics Module or the MEMS Module (which enhance the standard Solid Mechanics interface) is required.

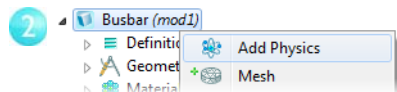
If you have a floating network license (FNL) or a class kit license (CKL) and your license file has been enabled for borrowing, you can borrow licenses from your license server. Select **Options>Licenses** from the main menu and then click **Borrow**. Select the licenses you want to borrow from the list and specify the number of days you want to keep them. Click **OK**.

If you want to add cooling by fluid flow, or don't have the Structural Mechanics Module or MEMS Module, read this section and then go to, "Fluid Flow" on page 51.

ADDING SOLID MECHANICS

1 Open the model `busbar.mph` that was created earlier. From the main menu, select **File>Save as** and rename the model `busbar_II.mph`.

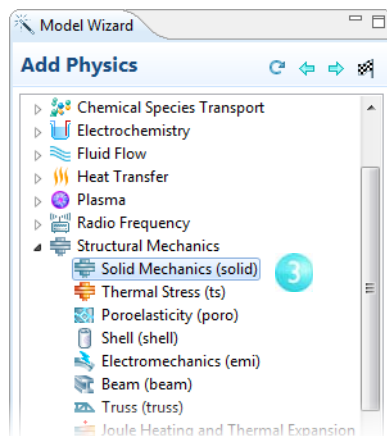
2 In the **Model Builder**, right-click the **Busbar** node  and select **Add Physics** .



- 3 In the **Model Wizard** under **Structural Mechanics**, select **Solid Mechanics**.

To add this interface, you can double-click it, right-click and select **Add Selected**, or click the **Add Selected** button.

- 4 Click the **Finish** button and save the file. You do not need to add any studies.

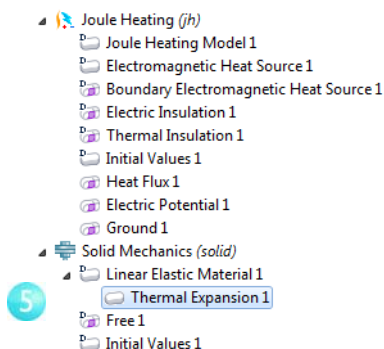


When adding additional physics, you need to make sure that materials included in the **Materials** node have all the required properties for the selected physics. In this example, you already know that all properties are available for copper and titanium.

You can start by adding the effect of thermal expansion to the structural analysis.

- 5 In the **Model Builder** under **Solid Mechanics**, right-click the **Linear Elastic Material 1** node and from the domain level, select **Thermal Expansion**.

A **Thermal Expansion** node is added to the **Model Builder**.



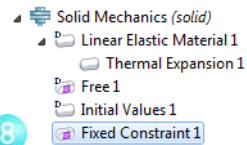
With the Structural Mechanics Module, the **Thermal Stress** predefined multiphysics interface is also available to define thermal stresses and strains.

- 6 In the **Thermal Expansion** settings window under **Model Inputs**, select **Temperature (jh/jhm1)** from the **Temperature** list.

This is the temperature field from the **Joule Heating** interface (**jh/jhm1**) and couples the Joule heating effect to the thermal expansion of the busbar.

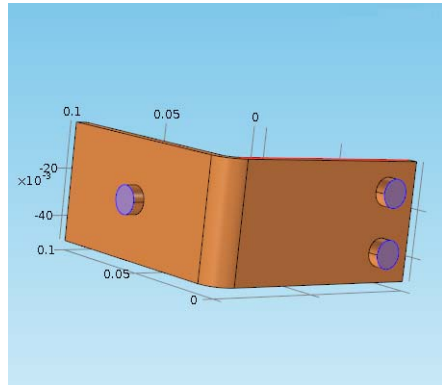
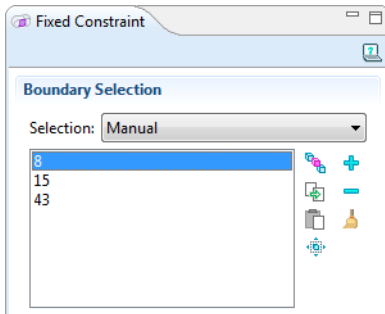
Next, fix the busbar at the position of the titanium bolts.

- 7 In the **Model Builder**, right-click **Solid Mechanics** and from the boundary level, select **Fixed Constraint**. A node with the same name is added to the **Model Builder**.



- 8 Click the **Fixed Constraint** node. In the **Graphics** window, rotate the busbar to view the back. Click one of the bolts to highlight it and right-click to add the bolt to the **Selection** list.
- 9 Repeat this procedure for the remaining bolts to add boundaries 8, 15, and 43.

- 9 Cross-check: Boundaries 8, 15, and 43.

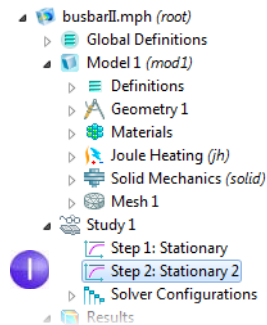


You can now update the Study to take the added effects into account.


RUNNING A STUDY SEQUENCE—JOULE HEATING AND THERMAL EXPANSION

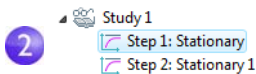
The Joule heating effect is independent of the stresses and strains in the busbar, assuming small deformations and ignoring the effects of electric contact pressure. This means that you can run the simulation using the temperature as input to the structural analysis. In other words, the extended multiphysics problem is weakly coupled. As such, you can solve it in two separate study steps—one for the strongly coupled Joule heating problem and a second one for the structural analysis.

- 1 In the **Model Builder**, right-click **Study 1**  and select **Study Steps>Stationary**  to add a second stationary study step.





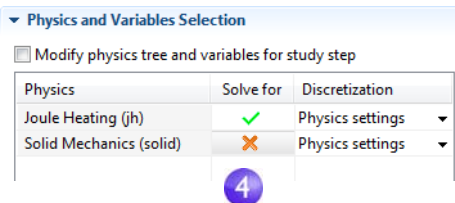
- 1 When adding study steps you need to manually connect the correct physics with the correct study step. Start by removing the structural analysis from the first step.

- 2 Under **Study 1**, click the **Step 1: Stationary** node .



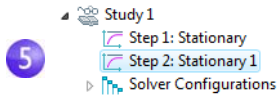
- 3 In the **Stationary** settings window, locate the **Physics and Variables Selection**.

- 4 In the **Solid Mechanics (solid)** row under **Solve for**, click to change the check mark  to an  to remove Solid Mechanics from Study 1.



Now repeat these steps to remove Joule heating from the second study step.

5 Under **Study 1**, click **Step 2: Stationary 1**.



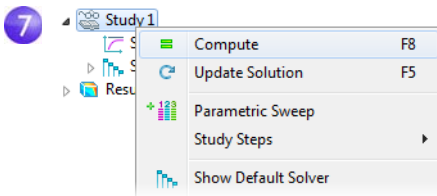
6 Under **Physics and Variables Selection**, in the **Joule Heating (jh)** row under **Solve for**, click to change the check mark ✓ to an ✗ to remove Joule Heating from Step 2.

6

A screenshot of the 'Physics and Variables Selection' dialog box. It has a checkbox 'Enable selection of variables and physics features' which is unchecked. Below it is a table with three columns: 'Physics interface', 'Solve for', and 'Discretization'.

Physics interface	Solve for	Discretization
Joule Heating (jh)	✗	Physics settings
Solid Mechanics (solid)	✓	Physics settings

7 Right-click the **Study 1** node and select **Compute** (or press F8) to solve the problem.

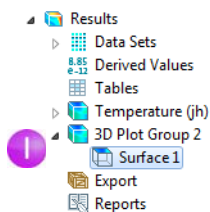



After a few moments of computation a second plot group is added under Results in the Model Builder. Save the file **busbarII.mph**, which now includes the **Solid Mechanics** interface and the additional study step.

RESULTING DEFORMATION

Now add a displacement to the plot.

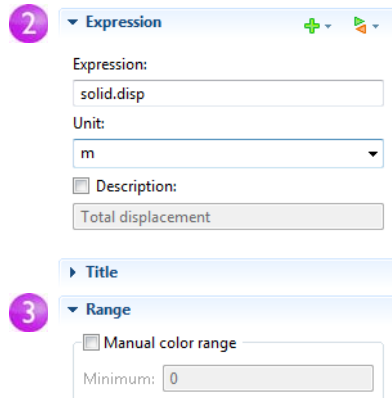
1 Under **Results>3D Plot Group 2**, click the **Surface 1** node.



- 2 In the **Surface** settings window in the **Expression** section, click the **Replace Expression** button .

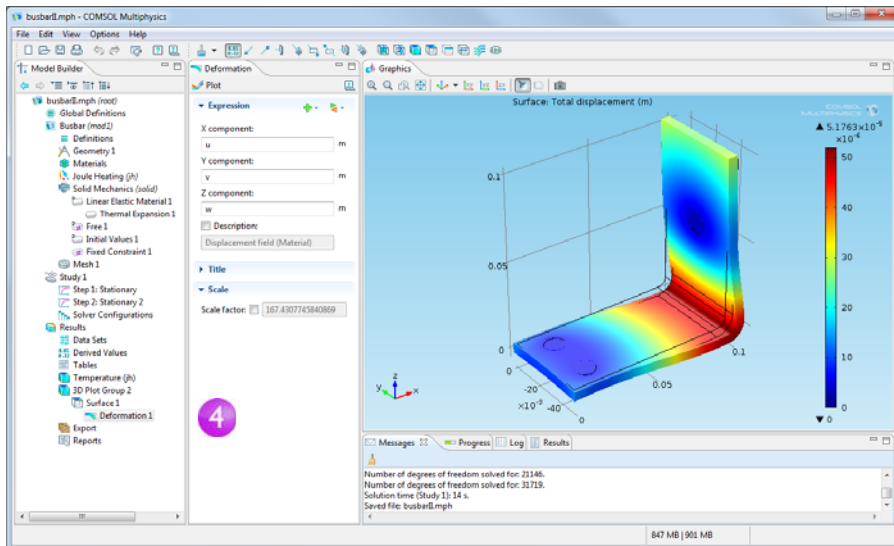
From the context menu, select **Solid Mechanics>Displacement>Total Displacements**. You can also enter `solid.disp` in the **Expression** field.

- 3 Click **Range** to expand the section. Click to clear the **Manual color range** check box.



The local displacement, due to thermal expansion, is displayed by COMSOL as a surface plot. Next add exaggerated deformation information.

- 4 In the **Model Builder**, under **Results>3D Plot Group 2**, right-click the **Surface 1** node and add a **Deformation**. The plot automatically updates in the **Graphics** window.




- 5 Save the `busbarII.mph` file, which now includes a **Surface** plot with a **Deformation**.

You can also plot the von Mises and principal stresses to assess the structural integrity of the busbar and the bolts.

FLUID FLOW

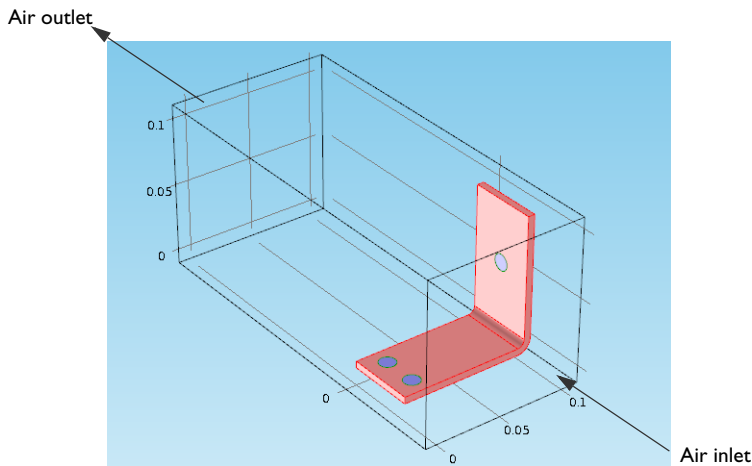
After analyzing the heat generated in the busbar and possibly the induced thermal stresses, you might want to investigate ways of cooling it by letting air flow over its surfaces. These steps do not require any additional modules.



When you have the CFD Module or the Heat Transfer Module, the **Conjugate Heat Transfer**  multiphysics interface is available. This automatically defines coupled heat transfer in solids and fluids including laminar or turbulent flow.

Adding fluid flow to the Joule heating model forms a new multiphysics coupling. To simulate the flow domain, you need to create an air box around the busbar. You can do this manually by altering the geometry from your first model or by opening a Model Library file.


Having loaded or created the geometry, now simulate the air flow as in this figure:.




DEFINING INLET VELOCITY


Start by adding a new parameter for the inlet flow velocity.




Select **View>Model Library**  and navigate to **COMSOL Multiphysics>Multiphysics>busbar_box**. Double-click to open the model file, which

contains the geometry in addition to the steps completed up to the end of the section “Customizing Materials” on page 41.

- 2 Under **Global Definitions**, click the **Parameters** node .

2  busbar_box.mph (root)

- Global Definitions
 - Parameters**

3  **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/(m^2·K)	
Vtot	20[mV]	0.020000 V	
Vin	1e-1[m/s]	0.10000 m/s	

- 3 In the **Parameters** settings window, click the last empty row in the **Name** column and enter Vin. Enter 1e-1 [m/s] in the **Expression** column and a description of your choice in the **Description** column.
- 4 Select **File>Save As** and save the model with a new name, busbar_box_I.mph.


ADDING AIR

The next step is to add the material properties of air.

- 1 Select **View>Material Browser**.
- 2 In the **Material Browser**, expand the **Built-In** tree. Right-click **Air** and select **+ Add Material to Model**. Click the **Material Browser** tab and close it.
- 3 In the **Model Builder** under **Materials**, click the **Air** node.

3  **Materials**

- Copper
- Titanium beta-215
- Air**

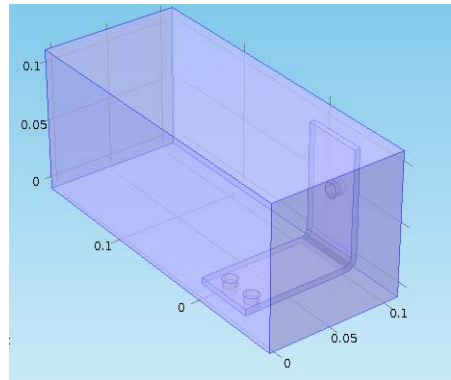
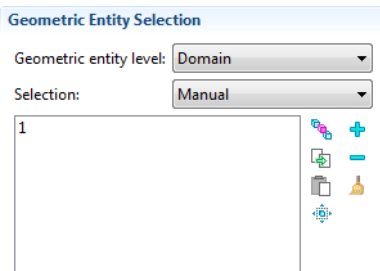
2  **Materials**

Search

- Built-In
 - Air**
 - + Add Material to Model
 - Remove Selected
 - Aluminum 6063-T83
 - Aluminum
 - American red oak
 - Beryllium copper UNS C17200

- 4 In the **Graphics** window, click the air box (Domain I) to highlight it (in red) and right-click to add it to the **Selection** list (which changes the color to blue).

4 Cross-check: Domain I

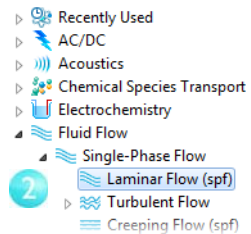




ADDING FLUID FLOW

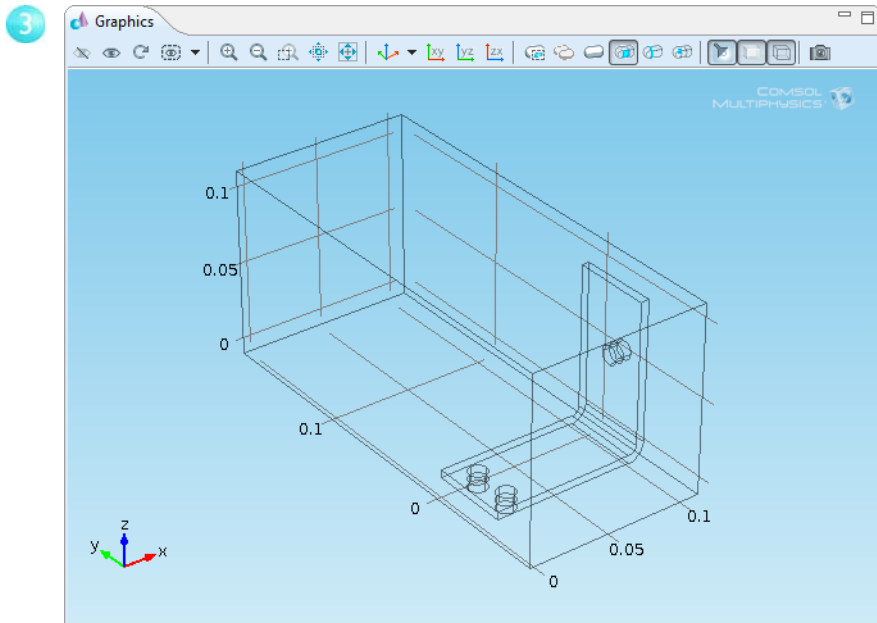
Now add the physics of fluid flow.

- 1 In the **Model Builder**, right-click **Model I** and select **Add Physics**.

- 2 In the **Add Physics** tree, under **Fluid Flow>Single-Phase Flow** double-click **Laminar Flow** to add it to the **Selected physics** section. You do not need to add any more studies. Click the **Finish** button.

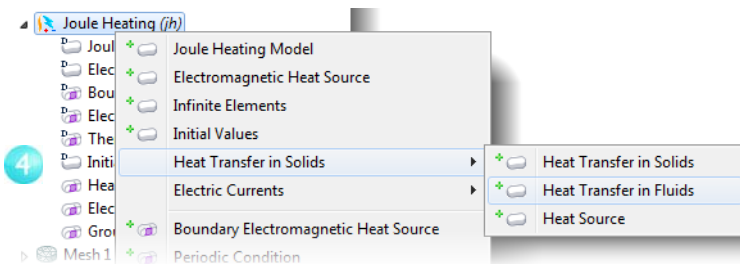


- 3 On the **Graphics** toolbar, click the **Select Boundaries** button  and then the **Wireframe** rendering button  to look inside the box.



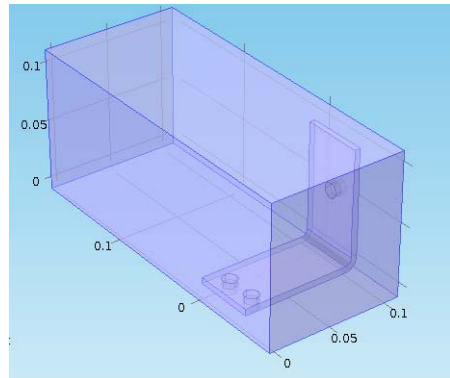
Now that you have added fluid flow to the model, you need to couple the heat transfer part of the **Joule Heating** interface to the fluid flow.

- 4 In the **Model Builder**, right-click **Joule Heating** . In the first section of the context menu, the domain level, select **Heat Transfer in Solids>Heat Transfer in Fluids**.



- 5 In the **Graphics** window select the air box (Domain 1) and right-click to add it to the **Selection** list.

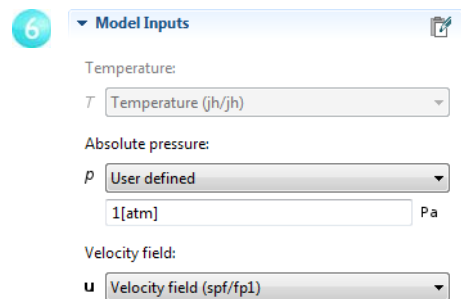
5 Cross-check: Domain 1.



Now couple fluid flow with heat transfer:

- 6 In the **Heat Transfer in Fluids** settings window under **Model Inputs**, select **Velocity field (spf/fp1)** from the **Velocity field** list.

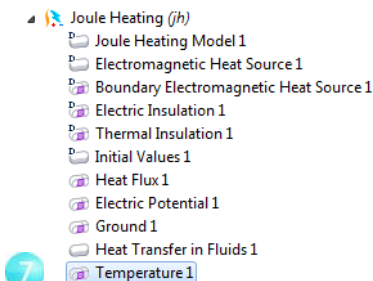
This identifies the flow field from the **Laminar Flow** interface and couples it to heat transfer:



Now define the boundary conditions by specifying the inlet and outlet for the heat transfer in the fluid domain.

- 7 In the **Model Builder**, right-click **Joule Heating**. In the second section of the context menu, the boundary section, select **Heat Transfer in Solids>Temperature**.

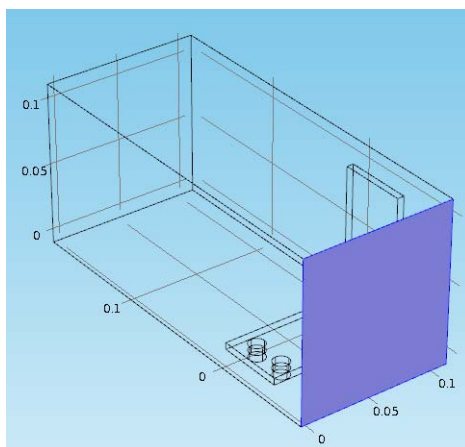
A **Temperature** node is added to the **Model Builder**



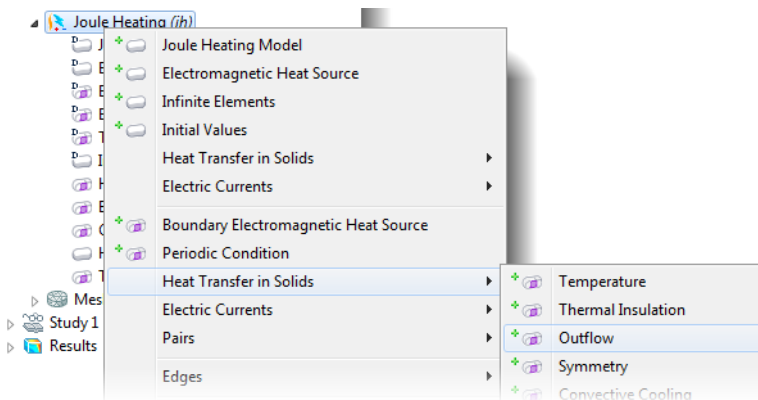
- 8 In the **Graphics** window, click the inlet boundary, Boundary 2, and right-click to add it to the **Selection** list.

This sets the inlet temperature to 293 K, the default setting. Continue by defining the outlet.

- 8 Cross-check: Boundary 2.



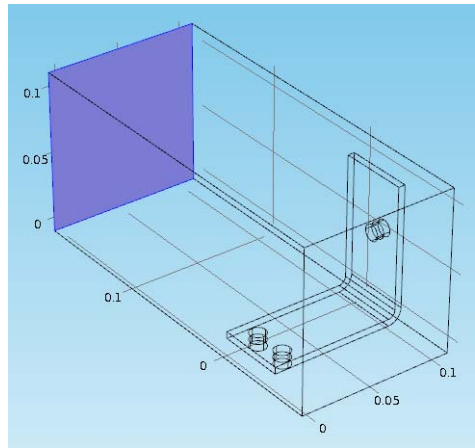
- 9 In the **Model Builder**, right-click **Joule Heating**. At the boundary level, select **Heat Transfer in Solids>Outflow**. An **Outflow** node is added to the **Model Builder**



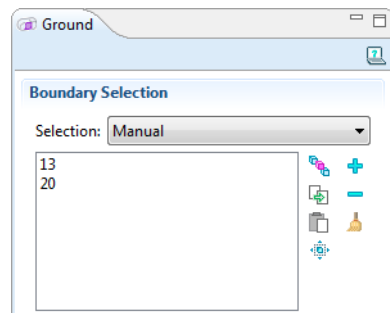
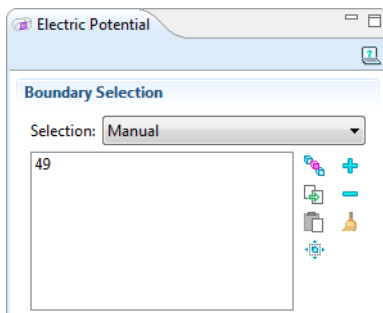
10 In the **Graphics** window, click the outlet boundary, Boundary 5, and right-click to add it to the **Selection** list.

10



Cross-check: Boundary 5.

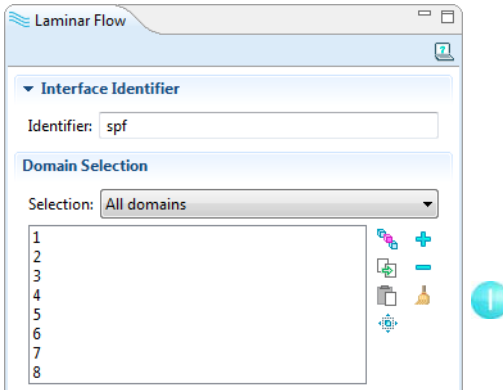


The settings for the busbar, the bolts, and the **Electric Potential I** and **Ground I** boundaries have retained the correct selection, even though you added the box geometry for the air domain. To confirm this, click the **Electric Potential I** and the **Ground I** nodes in the **Model Builder** to verify that they have the correct boundary selection.





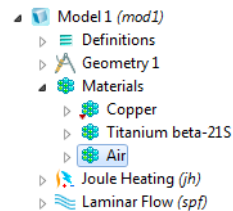
Continue with the flow settings. You need to indicate that fluid flow only takes place in the fluid domain and then set the inlet, outlet, and symmetry conditions.

- 1 In the **Model Builder**, click the **Laminar Flow** node . In the **Laminar Flow** settings window, click the **Clear Selection** button .



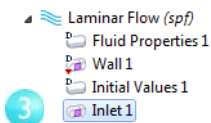
- 2 In the **Graphics** window click the air box (Domain 1) and right-click to add it to the **Selection**.

-  It is good practice to verify that the **Air** material under the **Materials** node has all the properties that this multiphysics combination requires. In the **Model Builder** under **Materials**, click **Air**. In the **Material** settings window under **Material Contents**, verify that there are no missing properties, which are marked with a warning sign . The section “Materials” on page 17 has more information.



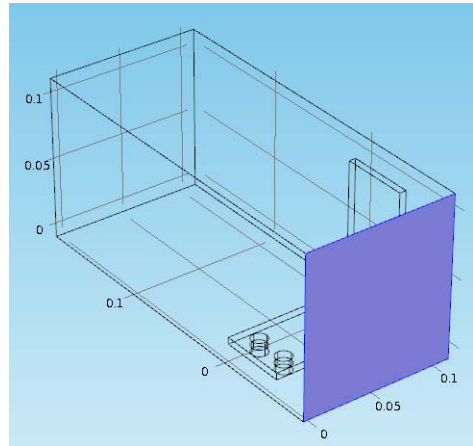
Continue with the boundaries.

- 3 In the **Model Builder**, right-click **Laminar Flow**  and at the boundary level select **Inlet**. An **Inlet** node  is added to the **Model Builder**.

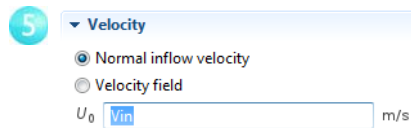


- 4 In the **Graphics** window, select the inlet (Boundary 2) and right-click to add it to the **Selection** list.

4 Cross-check: Boundary 2.

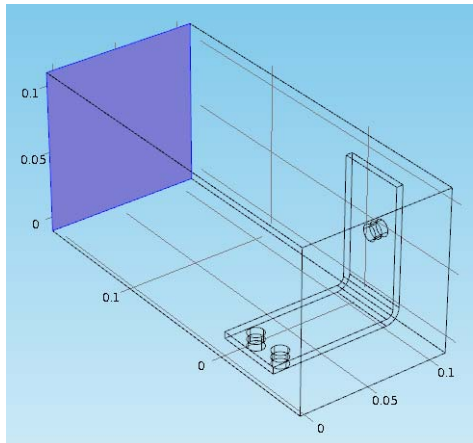


- 5 In the **Inlet** settings window under **Velocity** in the U_0 field, enter V_{in} to set the **Normal inflow velocity**.



- 6 Right-click **Laminar Flow** and at the boundary level select **Outlet**. In the **Graphics** window, select the outlet (Boundary 5) and right-click to add it to the **Selection** list.

6 Cross-check: Boundary 5.

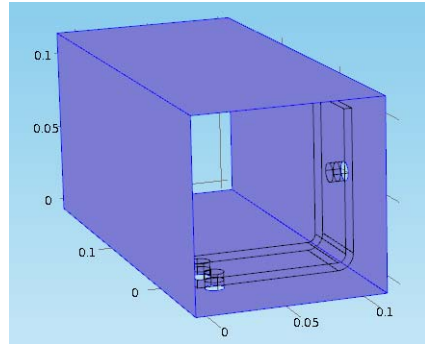


The last step is to add symmetry boundaries. You can assume that the flow just outside of the faces of the channel is similar to the flow just inside these faces. This is correctly expressed by the symmetry condition.

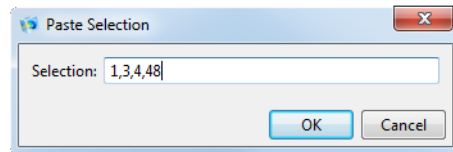
- 7 Right-click **Laminar Flow** and select **Symmetry**. A **Symmetry** node is added to the sequence.
- 8 In the **Graphics** window, click one of the blue faces in the figure (Boundaries 1, 3, 4, or 48) and right-click each one to add all to the **Selection** list.

Save the `busbar_box1.mph` file, which now includes the **Air** material and **Laminar Flow** interface settings.

8 Cross-check: Boundaries 1, 3, 4 and 48.



- When you know the boundaries, you can click the **Paste** button and enter the information. In this example, enter 1,3,4,48 in the **Paste selection** window. Click **OK** and the boundaries are automatically added to the **Selection** list.



The next step is to change the mesh slightly.

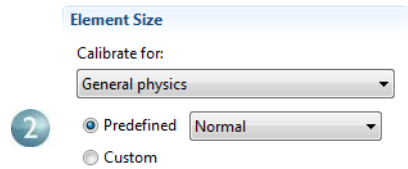
COARSENING THE MESH


In order to get a quick solution, the mesh is changed slightly and made coarser. The current mesh settings would take a relatively long time to solve, and you can always refine it later.

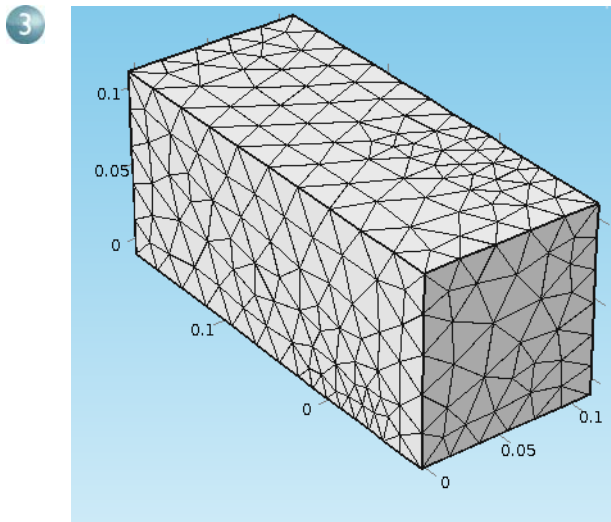
- In the **Model Builder**, expand the **Mesh 1** node and click the **Size** node.



- 2 In the **Size** settings window under **Element Size**, click the **Predefined** button and ensure that **Normal** is selected.



- 3 Click the **Build All** button . The geometry displays with a coarse mesh in the **Graphics** window.





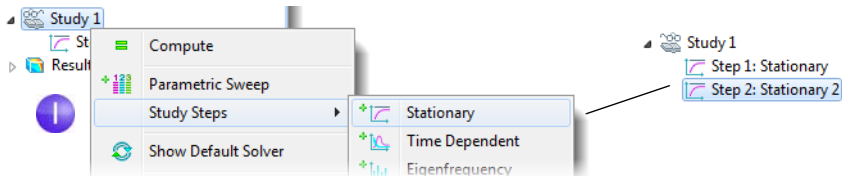
You can assume that the flow velocity is large enough to neglect the influence of the temperature increase in the flow field.

It follows that you can solve for the flow field first and then solve for the temperature using the results from the flow field as input. This is implemented with a study sequence.

RUNNING A STUDY SEQUENCE—FLUID FLOW AND JOULE HEATING

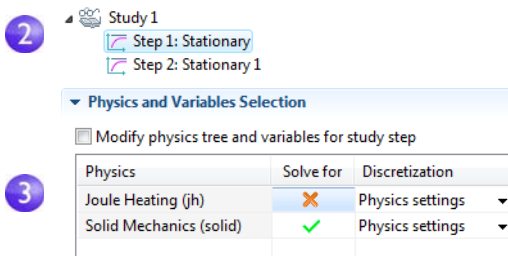
When the flow field is solved before the temperature field, it yields a weakly coupled multiphysics problem. The study sequence described in this section automatically solves such a weak coupling.






- 1 In the **Model Builder**, right-click **Study 1**  and select **Study Steps>Stationary**  to add a second stationary study step to the **Model Builder**.

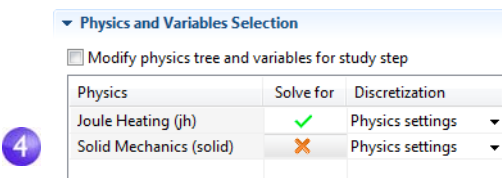




Next, the correct physics needs to be connected with the correct study step. Start by removing Joule heating from the first step.

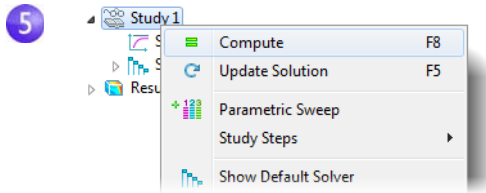
- 2 Under **Study 1**, click **Step 1: Stationary** .




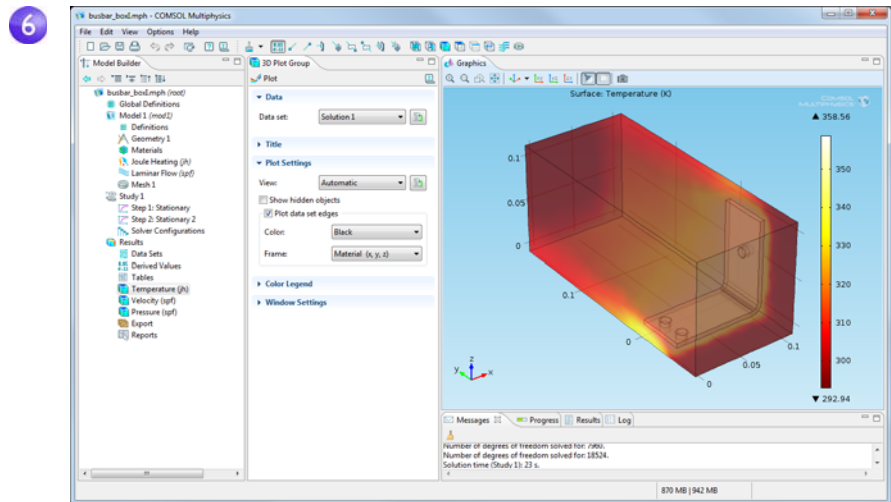
- 3 In the **Stationary** settings window locate the **Physics and Variables Selection** section. In the **Joule Heating (jh)** row, click to change the check mark  to an  in the **Solve for** column, removing **Joule Heating (jh)** from Study 1.
- 4 Repeat the step. Under **Study 1** click **Step 2: Stationary 1** . Under **Physics and Variables Selection**, in the **Laminar Flow (spf)** row click in the **Solve for** column to change the check mark  to an .



- 5 Right-click the **Study 1** node  and select **Compute**  (or press F8) to automatically create a new solver sequence that solves the two problems in sequence.



- 6 After the solution is complete, click the **Transparency** button  on the **Graphics** toolbar to visualize the temperature field inside the box.



The **Temperature Surface** plot that displays in the **Graphics** window shows the temperature in the busbar and in the surrounding box. You can also see that the temperature field is not smooth due to the relatively coarse mesh. A good strategy to get a smoother solution would be to refine the mesh to estimate the accuracy.

- 7 Save the **busbar_box_1.mph** file up to this point so you can return to this file if you want. The next steps use the original **busbar.mph** file.

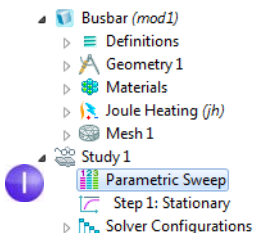
Parameter Sweeps

Sweeping a Geometric Parameter

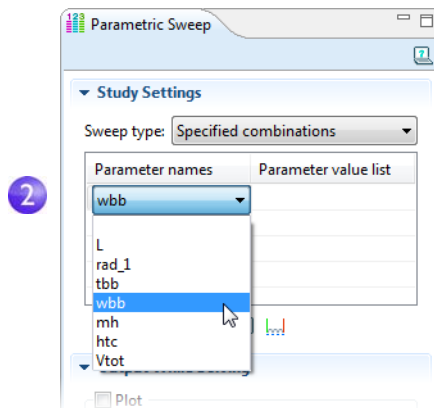
Often it is interesting to generate multiple instances of a design to meet specific constraints. For the busbar, a design goal might be to lower the operating temperature and a decrease in the current density achieves this. Since the current density depends on the geometry of the busbar, varying the width, **wbb**, should change the current density and, in turn, have some impact on the operating temperature. Run a parametric sweep on **wbb** to study this change.


ADDING A PARAMETRIC SWEEP

- 1 Open the model file **busbar.mph**. In the **Model Builder**, right-click **Study 1** and select **Parametric Sweep**. A **Parametric Sweep** node is added to the **Model Builder** sequence.

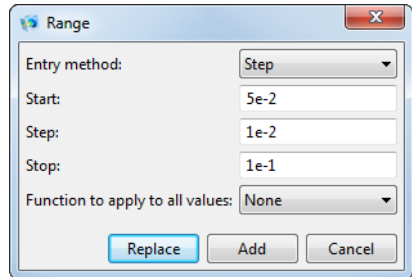


- 2 In the **Parametric Sweep** settings window, under the table, click the **Add** button. From the **Parameter names** list in the table, select **wbb**.





- 3 Enter a range of **Parameter values** to sweep the width of the busbar from 5 cm to 10 cm with 1 cm increments. There are different ways to enter this information:
 - Copy and paste or enter range(0.05,0.01,0.1) into the **Parameter value list** field.
 - Click the **Range**  button and enter the values in the **Range** dialog box. In the **Start** field, enter 5e-2. In the **Step** field, enter 1e-2 and in the **Stop** field, enter 1e-1.
 - Click **Browse** to **Load parameter values** from a text file.

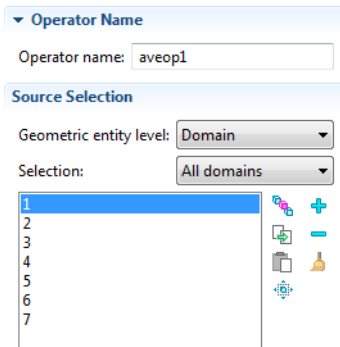
3





Next an Average Model Coupling is created, which can later be used to calculate the average temperature in the busbar.

- 4 Under **Model I**, right-click **Definitions**  and select **Model Couplings>Average** .
- 5 In the **Average** settings window select **All domains** from the **Selection** list. This creates an operator called **aveop1**.

5



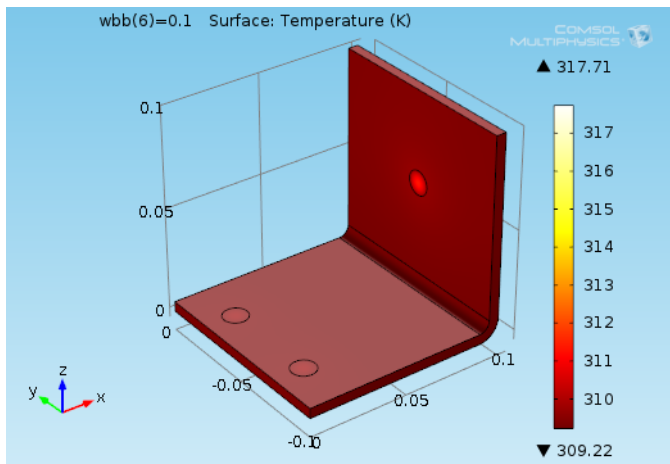
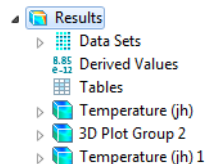
The **aveop1** is now available to calculate the average of any quantity defined on those domains. A little later this is used to calculate the average temperature, but it can also be used to calculate average electric potential, current density, and so forth.


- 6 Right-click **Study I**  and select **Compute**  to run the sweep.
- 7 Select **File>Save As** to save the model with a new name, **busbar_III.mph**.

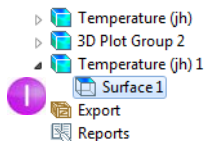
PARAMETRIC SWEEP RESULTS

A **Temperature (jh)** node is added under **Results**.

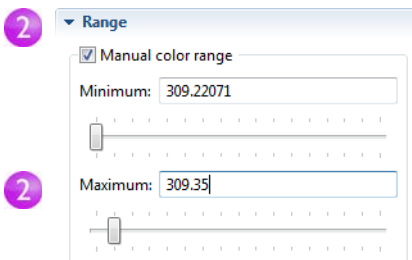
The plot that displays in the **Graphics** window after the parametric sweep shows the temperature in the wider busbar using the last parameter value, $wbb=0.1$ m (10 cm). The plot is uniform in color, so change the maximum color range.



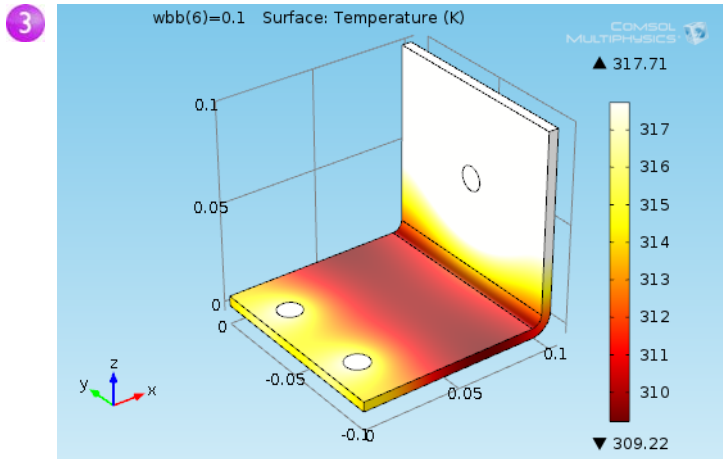
1 Under the **Temperature (jh)** node, click the **Surface** node .



2 In the **Surface** settings window, click **Range** to expand the section. Select the **Manual color range** check box. Enter **309.35** in the **Maximum** field (replace the default) to plot wbb at 10 cm.

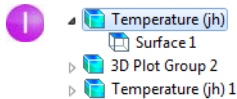


- 3 The **Temperature (jh)** plot is updated in the **Graphics** window for $wbb = 0.1$ m (10 cm).



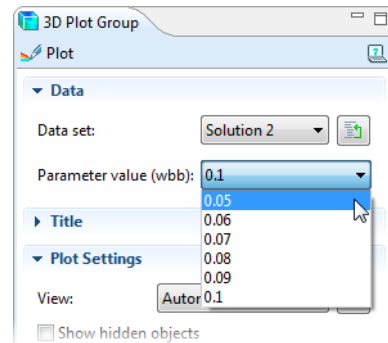
Compare the wider busbar plot to the temperature for $wbb = 0.05$ m (5 cm).

- 1 In the **Model Builder**, click the first **Temperature (jh)** node.

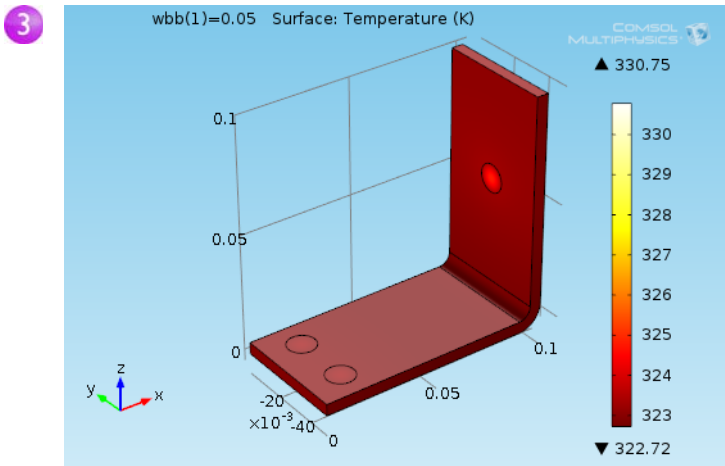


- 2 In the **3D Plot Group** settings window, select **Solution 2** from the **Data set** list. This data set contains the results from the parametric sweep.


- 3 In the **Parameters value** list, select **0.05** (which represents $wbb = 5$ cm). Click the **Plot** button. Click the **Zoom Extents** button on the **Graphics** toolbar.



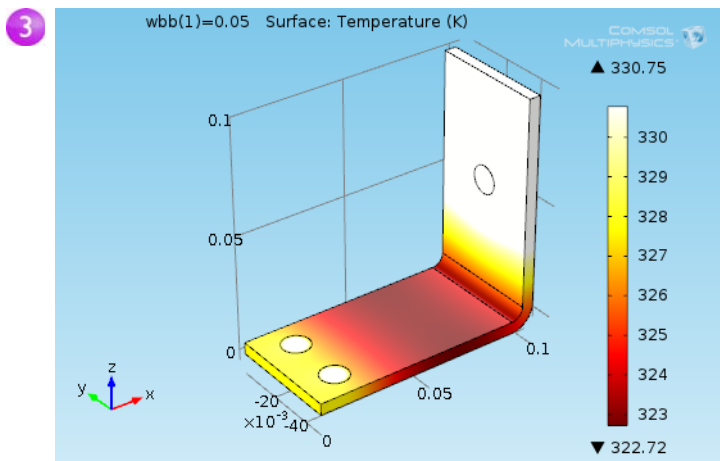
The **Temperature (jh)** plot is updated for $wbb = 0.05 \text{ m}$ (5 cm).



Like the wider busbar, the plot is uniform in color, so change the maximum color range.

- 1 Under the first **Temperature (jh)** node, click the **Surface** node .
- 2 In the **Surface** settings window, click **Range** to expand the section (if it is not already expanded). Select the **Manual color range** check box.
- 3 Enter 323 in the **Maximum** field (replace the default) to plot wbb at 5 cm.

The **Temperature (jh)** plot is updated in the **Graphics** window for $wbb = 0.05 \text{ m}$ (5 cm).

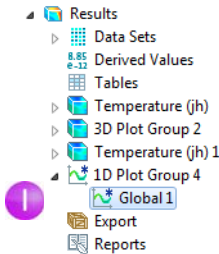


Click the first and second **Temperature** plot nodes to compare the plots in the **Graphics** window. It shows that the maximum temperature decreases from 331 K to 318 K as the width of the busbar increases from 10 cm to 5 cm.

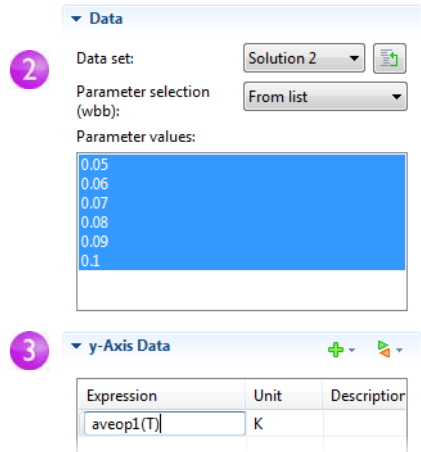
ADDING MORE PLOTS


To further analyze these results, you can plot the average temperature for each width.

- 1 Right-click **Results** and add a **ID Plot Group**. Then right-click **ID Plot Group 4** and add a **Global** node to the **Model Builder**.

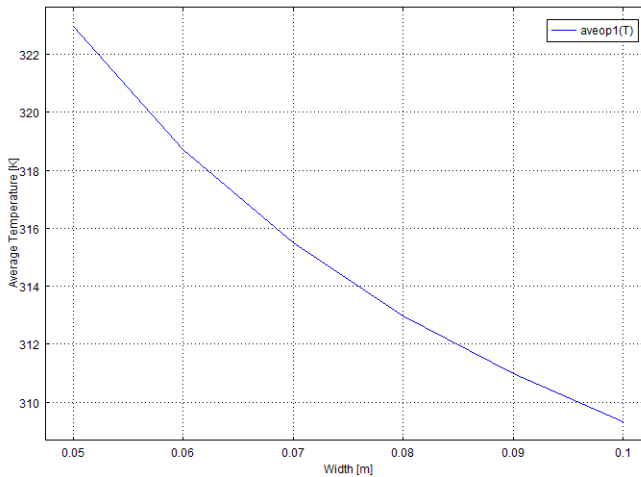


- 2 In the **Global** settings window, select **Solution 2** from the **Data set** list (the parametric sweep results). Select **From list** as the **Parameter selection (wbb)**.
- 3 Under **y-Axis Data**, click the first row in the **Expressions** column and enter **aveop1(T)**. You use a similar syntax to calculate the average of other quantities.
- 4 Click to expand the **Title** section. Select the **Expression** check box.



- 5 Click the **Plot** button  and save the **busbar_III.mph** model with these additional plots that use the parametric sweep results.

3



In the plot, the average temperature also decreases as the width increases. This indicates that the goal of a lower operating temperature would be fulfilled by using a wider busbar. Of course, this may also increase the total mass (and therefore the cost) of the busbar. This suggests an optimization problem for you to consider.

The subject of parameter sweeps naturally raises the question of parallel processing; it would be efficient if all parameters were solved simultaneously.

Parallel Computing

COMSOL supports most forms of parallel computing including shared memory parallelism (for example, multicore processors) and high performance computing (HPC) clusters.

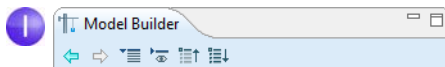
You can use clusters to solve a series of parameter steps for a model, one parameter per node, or you can solve a single large model using distributed memory. For maximum performance, the COMSOL cluster implementation can utilize shared-memory multicore processing on each node in combination with the MPI-based distributed memory model. This brings a major performance boost by making the most out of the computational power installed.



ADDING A CLUSTER COMPUTING JOB

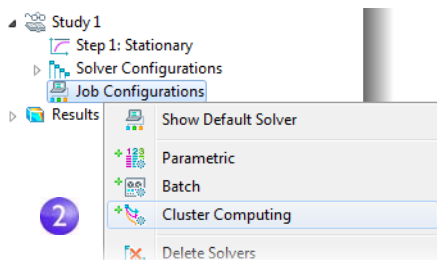
Continue these steps using the `busbar_III.mph` model created in “Adding a Parametric Sweep” on page 64.

To run cluster simulations, you need to first enable the advanced options for the **Job Configurations** node.

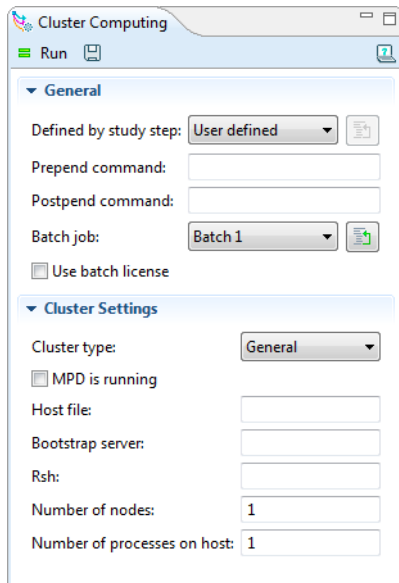
- 1 Click the **Show** button  on the **Model Builder** and select **Advanced Study Options**.



- 2 In the **Model Builder**, under **Study 1**, right-click **Job Configurations**  and select **Cluster Computing** .



The **Cluster Computing** settings window helps to manage the simulation either for running several instances of an identical parameterized model – one parameter value per host – or for running one parameter step in distributed mode.



The screenshot shows the 'Cluster Computing' dialog box with two main sections: 'General' and 'Cluster Settings'. In the 'General' section, 'Defined by study step' is set to 'User defined', 'Batch job' is 'Batch 1', and 'Use batch license' is unchecked. The 'Cluster Settings' section shows 'Cluster type' as 'General', 'MPD is running' is unchecked, and the 'Number of nodes' and 'Number of processes on host' are both set to 1. Other fields like 'Prepend command', 'Postpend command', 'Host file', 'Bootstrap server', and 'Rsh' are empty.

Section	Property	Value
General	Defined by study step	User defined
	Prepend command	
	Postpend command	
	Batch job	Batch 1
	Use batch license	<input type="checkbox"/>
Cluster Settings	Cluster type	General
	MPD is running	<input type="checkbox"/>
	Host file	
	Bootstrap server	
	Rsh	
	Number of nodes	1
	Number of processes on host	1

You choose the type of cluster job you want to do from the **Cluster type** list. COMSOL supports Windows Computer Cluster Server (WCCS) 2003, Windows HPC Server (HPCS) 2008, Open Grid Scheduler/ Grid Engine (OGS/GE), or Not distributed.

To learn more about running COMSOL in parallel, see the *COMSOL Installation and Operations Guide*.

Geometry Sequences

This section details how to create the busbar geometry using COMSOL's geometry tools. The step-by-step instructions take you through the construction of the geometry using parameters set up in Global Definitions. Using parameterized dimensions helps to produce *what-if* analyses and automatic geometric parametric sweeps.

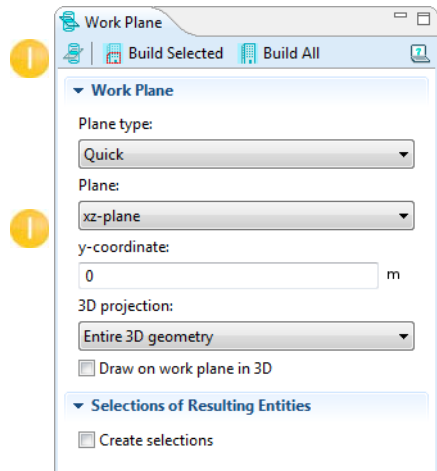
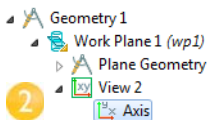
Follow the steps under the Model Wizard (to add the physics and study) and Global Definitions (to add the parameters) starting with “Thorough Example: The Busbar” on page 9. Then return to this section to learn about geometry modeling. The first step in the geometry sequence is to draw the profile of the busbar.

1 Under **Model I**, right-click **Geometry I** and select **Work Plane**. In the **Work Plane** settings window:

- Select **xz-plane** from the **Plane** list.
- Click the **Show Work Plane** button on the settings toolbar.

Continue by editing the axis and grid settings in Work Plane 1.

2 In the **Model Builder**, expand the **View 2** node and click **Axis**.



3 In the **Axis** settings window:

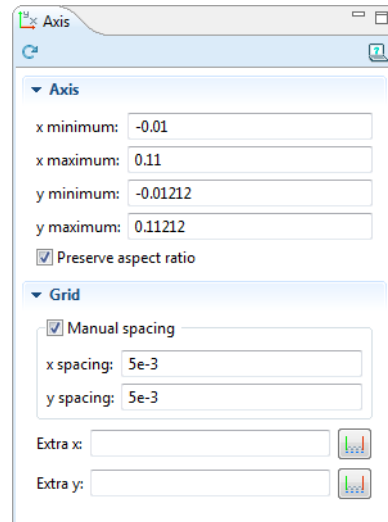
Under **Axis**:


- In the **x minimum** and **y minimum** fields, enter $-1e-2$. Replace the defaults.
- In the **x maximum** and **y maximum** fields, enter 0.11 . Replace the defaults.

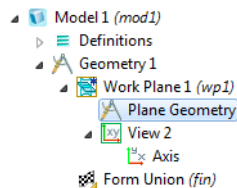
Under **Grid**:

- Select the **Manual Spacing** check box.
- In the **x spacing** and **y spacing** fields, enter $5e-3$.

4 Click the **Apply** button  on the toolbar:



You can use interactive drawing to create a geometry using the drawing toolbar buttons while pointing and clicking in the Graphics window. You can also right-click the **Plane Geometry** node  under **Work Plane I**  to add geometry objects to the geometry sequence.



Right-click the Geometry node to add geometry objects.

Click the Plane Geometry node under a Work Plane node to open the interactive toolbar:



In the next set of steps, a profile of the busbar is created.

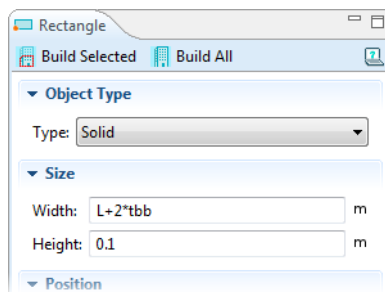
- 5 In the **Model Builder** under **Work Plane 1**, right-click **Plane Geometry** and select **Rectangle**.

In the **Rectangle** settings window under **Size** enter:

- $L+2*t_{bb}$ in the **Width** field
- 0.1 in the **Height** field

Click the **Build Selected** button.

5



- 6 Create a second rectangle. Under **Work Plane 1**, right-click **Plane Geometry** and select **Rectangle**.

Under **Size**, enter:

- $L+t_{bb}$ in the **Width** field
- $0.1-t_{bb}$ in the **Height** field

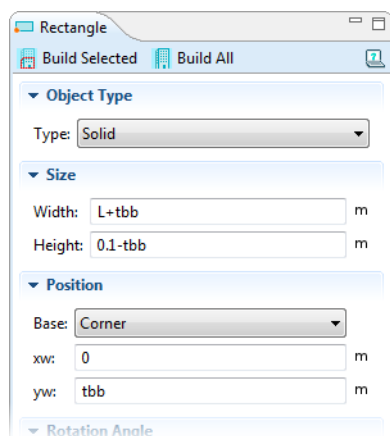
Under **Position**, enter:

- t_{bb} in the **yw** field

Click the **Build Selected** button.

6


6



Use the Boolean Difference operation to subtract the second rectangle from the first one.

- 7 Under **Work Plane 1**, right-click **Plane Geometry** and select **Boolean Operations>Difference**. In the **Graphics** window, click r_1 and right-click to add r_1 to the **Objects to add** list in the **Difference** settings window.

Note: To help select the geometry, you can display geometry labels in the **Graphics** window. In the **Model Builder** under **Geometry 1>WorkPlane 1>Plane Geometry**, click the **View 2** node. Go to the **View** settings window and select the **Show geometry labels** check box.

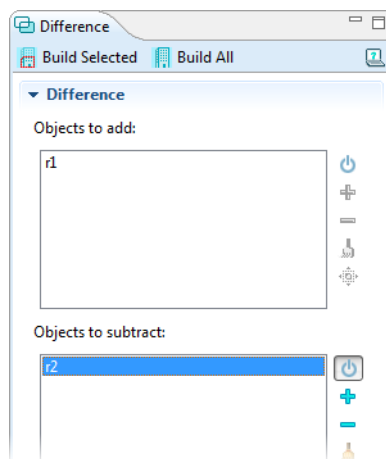
- 8 In the **Difference** settings window, click the **Activate selection** button  to the right of the **Objects to subtract** list. Then right-click to add r2 to the list.

Click **Build Selected** .

Note: Another way to select r2 in the **Graphics** window is to use the Selection List window. Select **View>Selection List** and in the **Selection List** settings window, click to highlight r2 (**solid**). Then right-click r2 (**solid**) in the list to add it to the **Object to subtract** list.

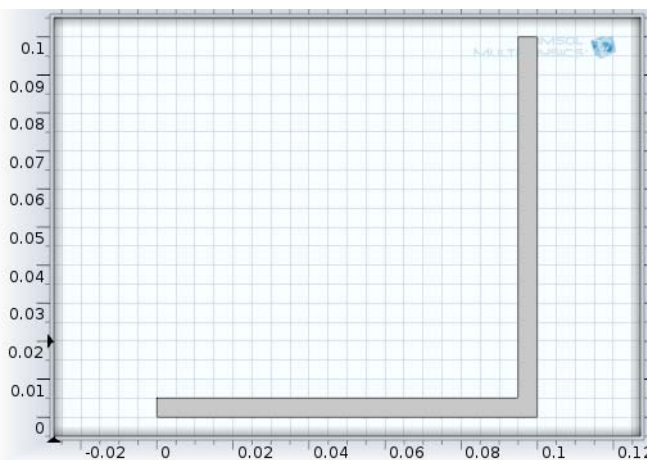
7

8



After building the selected geometry, you should have a backward-facing, L-shaped profile.


8

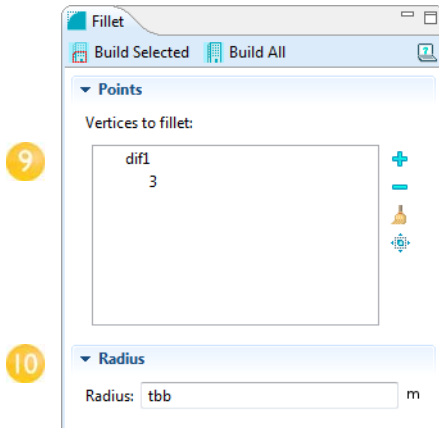


Continue by rounding the corners using fillets.

- 9 Under **Work Plane 1**, right-click **Plane Geometry**  and select **Fillet** .



Add point 3 to add it to the **Vertices to fillet** list. There are different ways to add the points:

- In the **Graphics** window, click point 3 (in the inner right corner), and right-click to add it to the **Vertices to fillet** list.
- Select **View>Selection List**. In the **Selection List** window, click 3. The corresponding point is highlighted in the **Graphics** window. Click the **Add to Selection** button  on the **Fillet** settings window, or right-click in the **Selection List**.



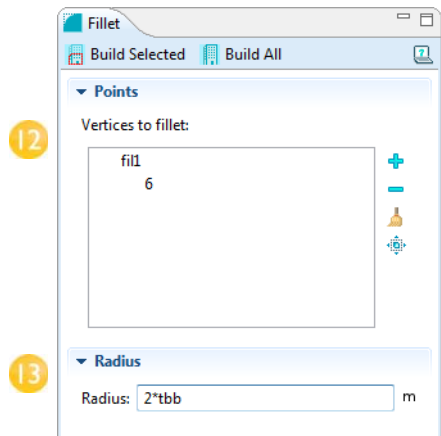
- 10 Enter **tbb** in the **Radius** field. Click **Build Selected** .

This takes care of the inner corner.

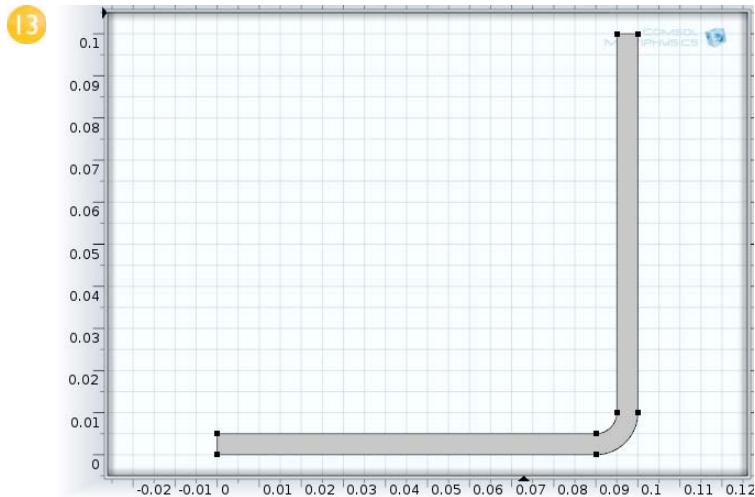
- 11 For the outer corner, right-click **Plane Geometry**  and select **Fillet** .

- 12 In the **Graphics** window, click point 6, the outer corner, and right-click to add it to the **Vertices to fillet** list.

- 13 Enter $2*tbb$ in the **Radius** field. Click **Build Selected** .

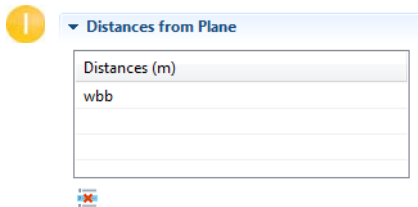


The result should match this figure:



Next you extrude the work plane to create the 3D busbar geometry.

- In the **Model Builder**, right-click **Work Plane 1** and select **Extrude**. In the **Extrude** settings window, enter **wbb** in the **Distances from Plane** table (replace the default) to extrude to the width of the profile.



The table allows you to enter several values in order to create sandwich structures with different layer materials. In this case, only one extruded layer is needed.

- 2



3

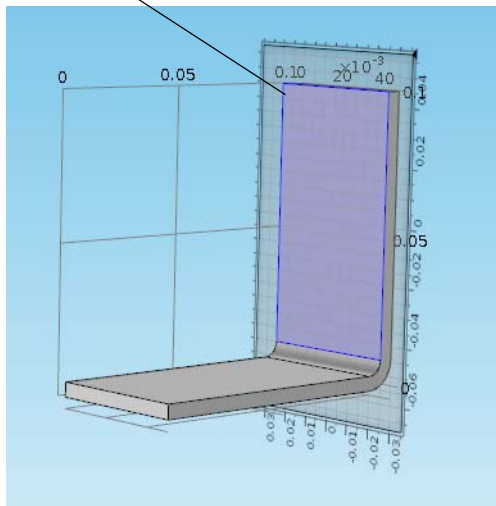
-





- 4 In the **Graphics** window, click face 8 (highlighted in the figure). Once this surface is highlighted in red, right-click anywhere in the **Graphics** window to add it to the **Planar face** list in the **Work Plane** settings window.

Face number 8 is now highlighted in blue and the work plane is positioned on top of face number 8.

4 Face 8



- 5 Click the **Show Work Plane** button  to draw the first circle representing the position of the first bolt. Click the **Zoom Extents** button  on the **Graphics** toolbar.

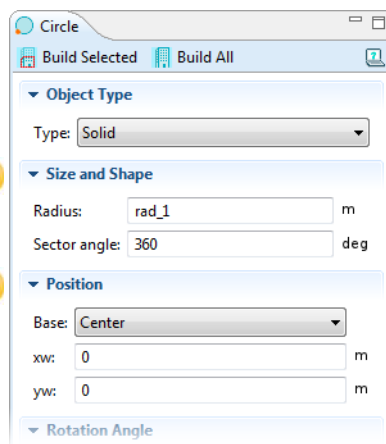
- 6 Under **Work Plane 2**, right-click **Plane Geometry**  and select **Circle** .

In the **Circle** settings window:

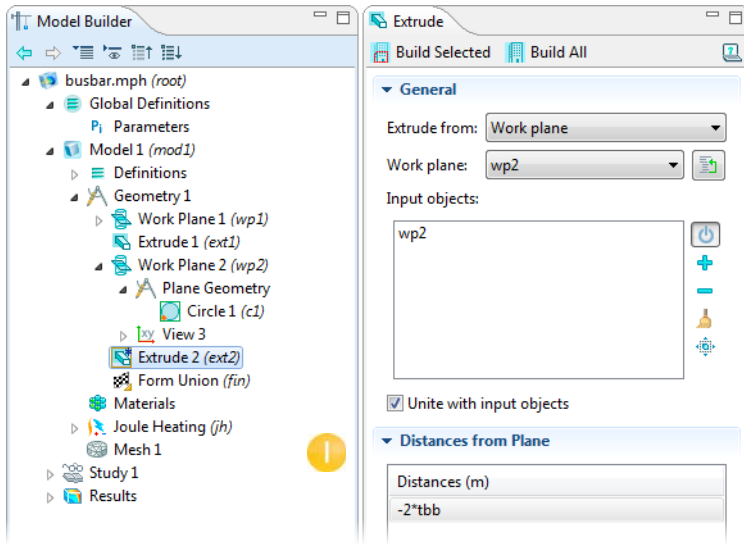
- Under **Size and Shape** in the **Radius** field, enter **rad_1**.
- Under **Position**, the **xw** and **yw** coordinates (0, 0) are already defined.

Click **Build Selected** .

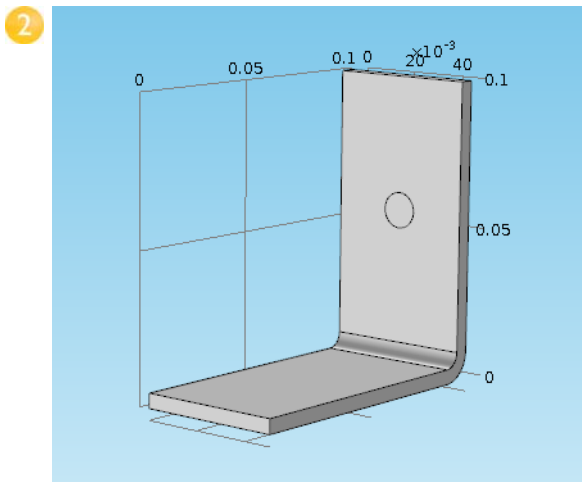
Continue creating the bolt by adding an extrude operation.



- In the **Model Builder**, right-click **Work Plane 2** and select **Extrude**. In the **Extrude** settings window, in the first row of the **Distances from Work Plane** table, enter $-2 \cdot t_{bb}$ to extrude the circle a distance equal to the thickness of the busbar:



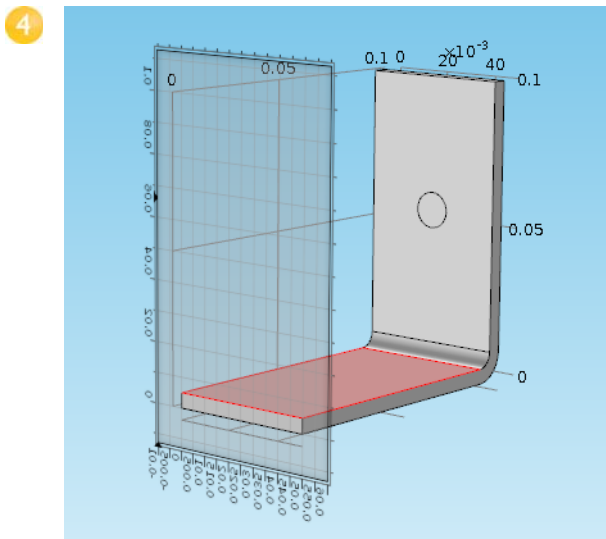
- Click the **Build Selected** button to create the cylindrical part of the titanium bolt that runs through the busbar:





Draw the two remaining bolts.

- Right-click **Geometry 1** and select **Work Plane**. A **Work Plane 3** node is added. In the **Work Plane** settings window, for **Work Plane 3**, select **Face parallel** as the **Plane type**.

- 4 In the **Graphics** window, click Face 6 (shown in the figure). When this surface is highlighted red, right-click anywhere in the **Graphics** window to add it to the **Planar face** list in the **Work Plane** settings window.



- 5 Click the **Show Work Plane** button  on the **Work Plane** settings window and the **Zoom Extents** button  on the **Graphics** toolbar to get a better view of the geometry.

To parameterize the position of the two remaining bolts, add the circles that form the cross sections of the bolts.



- 6 Under **Work Plane 3**, right-click **Plane Geometry**  and select **Circle** .

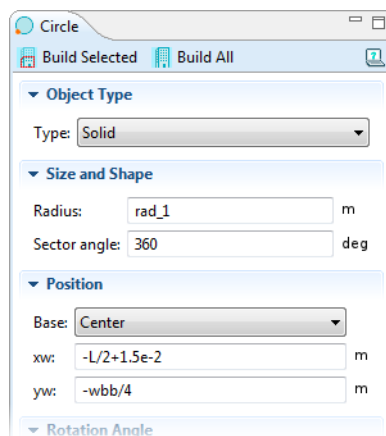
In the **Circle** settings window:


- Under **Size and Shape**, enter rad_1 in the **Radius** field.
- Under **Position**, enter $-L/2 + 1.5e-2$ in the **xw** field and $-wbb/4$ in the **yw** field.

Click **Build Selected** .

Copy the circle that you just created to generate the third bolt in the busbar.

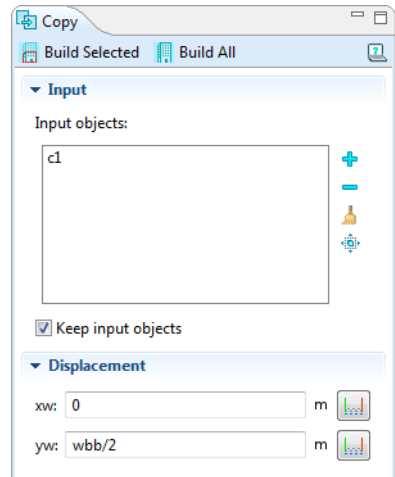
- 7 Under **Work Plane 3**, right-click **Plane Geometry**  and select **Transforms>Copy** .



- 8 In the **Graphics** window, click the circle **c1** to highlight it. Right-click anywhere in the **Graphics** window and add the circle to the **Input object** list in the **Copy** settings window.
- 9 In the **Copy** settings window under **Displacement**, enter $wbb/2$ in the **yw** field.
- 10 Click **Build Selected** .

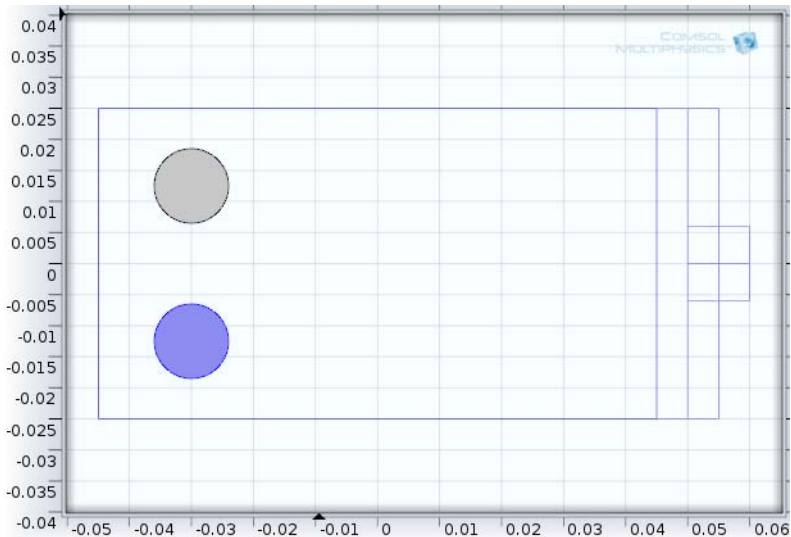
8

9






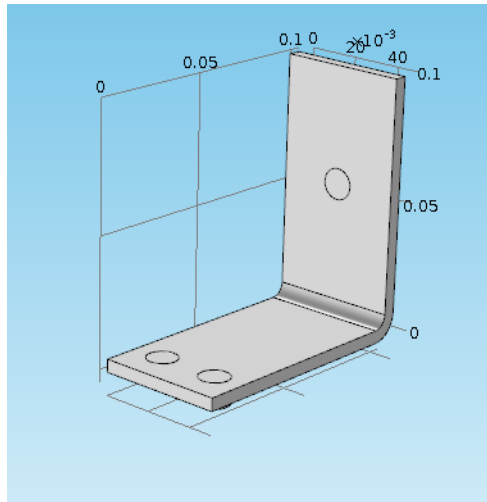
Your geometry, as shown in the workplane, should match this figure so far.

10



Continue by extruding the circles.

- 11 In the **Model Builder**, right-click **Work Plane 3**  and select **Extrude** . In the **Extrude** settings window, in the first row of the **Distances from Work Plane** table, enter $-2*tbb$ (replace the default). Click **Build All** .



Easy Example: The Wrench

Torquing a Wrench

Working with hand tools gives you a practical introduction to basics of engineering. At some point in your life It is likely you have tightened a bolt using a wrench. This exercise takes you through a model that analyzes this basic task, going into detail about the associated geometric tolerances, torque specs, and the intrinsic contact problem.

In principle, a bolt replaces the need for external fastening by providing an internal clamping force via pretension. The tensile stress in a bolt is produced when it is torqued into a matching threaded hole or nut. The magnitude of this stress depends on many factors, such as material selection, assembly configuration, and lubrication.


Control of these parameters becomes the focus of much engineering effort, especially in critical applications such as automotive engines, brakes, aircraft, and structural installations. The following model presents both a bolt and a wrench at the moment of installation.


The following tutorial is a short introduction to using COMSOL Multiphysics and it includes the basics of COMSOL Multiphysics. Start by opening the Model Wizard and add a physics and a study, then import a geometry, open the Material Browser to add a material and examine the material properties. The other key steps to creating a model are then explored by defining a parameter, defining the boundary conditions, selecting geometric entities in the Graphics window, defining the Mesh and Study nodes, and finally examining the resulting plots.

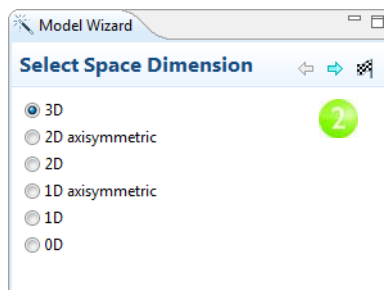
If you prefer to practice with a more advanced model, read this section to familiarize yourself with some of the key features, and then go to the tutorial “Thorough Example: The Busbar” on page 9.

MODEL WIZARD


1 Open the **Model Wizard**. To open the Model Wizard, double-click the COMSOL icon on the desktop. Or when COMSOL is already open:


- Click the **New** button  on the main toolbar
- Select **File>New** from the main menu
- Right-click the root node and select **Add Model**

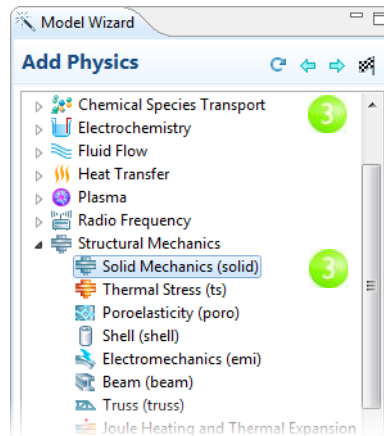
2 In the **Select Space Dimension** window that opens, the **3D** button is selected by default. Click **Next** .



3 In **Add Physics**, select **Structural Mechanics > Solid Mechanics (solid)** .


Double-click or right-click to add it to the **Selected physics** section. Click **Next** .

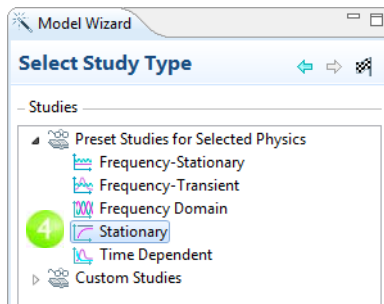
Another way to open the **Add Physics** window is to right-click the **Model** node and select **Add Physics** .



4 Click **Stationary**  under **Preset Studies**. Click the **Finish** button .

Preset Studies are studies that have solver and equation settings adapted to the selected physics, in this example, Joule heating.

Any selection from the **Custom Studies** branch  needs manual fine-tuning.



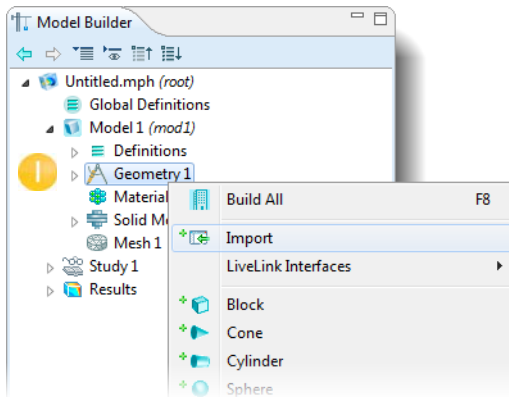
GEOMETRY

This tutorial uses a predefined geometry. To learn how to build your own geometry for the busbar model, see “Geometry Sequences” on page 73.

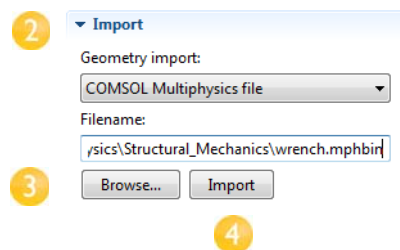
File Locations

For both the wrench and busbar tutorials in this guide, you can load predefined geometry or parameter files into COMSOL instead of manually adding the information. The location of the text and model files used in this exercise varies based on the software installation. For example, if the installation is on your hard drive, the file path might be similar to `C:\Program Files\COMSOL43\models\`.

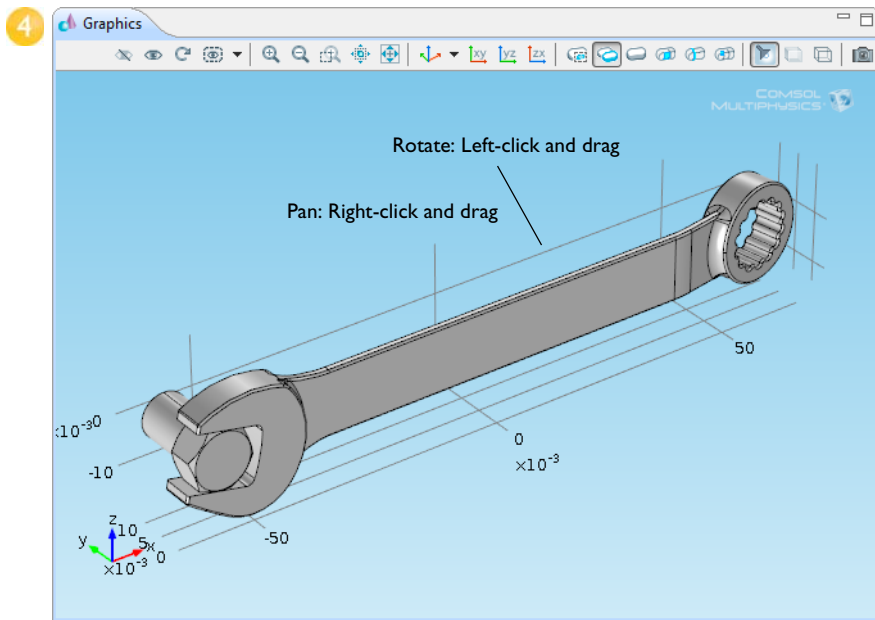
- 1 Under **Model 1**, right-click **Geometry 1** and select **Import**.









- 2 In the **Import** settings window, from the **Geometry import** list, select **COMSOL Multiphysics file**.
- 3 Click **Browse** and locate the file `wrench.mphbin` in the Model Library folder **COMSOL_Multiphysics/Structural_Mechanics**. Double-click to add or click **Open**.



4 Click **Import** to display the geometry in the **Graphics** window.




5 Click the **Save** button  on the main toolbar or select **File>Save** and name the model `wrench.mph`. Click the wrench geometry in the **Graphics** window and experiment with moving it around. As you click and right-click the geometry, it changes color. Click the **Zoom In** , **Zoom Out** , Go to **Default 3D View** , **Zoom Extents** , and **Transparency**  buttons on the toolbar to see what happens to the geometry.

- To rotate the busbar, left-click and drag it in the **Graphics** window
- To move it, right-click and drag
- To zoom in and out, center-click (and hold) and drag



Also see “Keyboard Shortcuts” on page I07 and “Working with 3D Geometry in the Graphics Window” on page I05 for additional information.

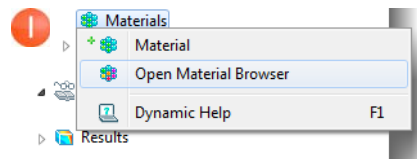
MATERIALS

The Materials node  stores the material properties for all physics and all domains in a Model node. The chosen bolt material and tool steel are important characteristics of this contact problem. Here is how to choose them in COMSOL.

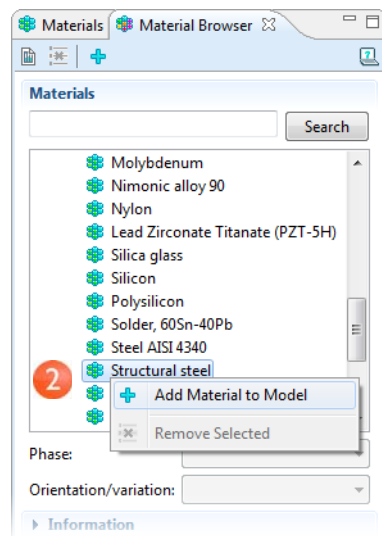
1 Open the **Material Browser**.

To open the Material Browser you can:

- Right-click **Materials**  in the **Model Builder** and select **Open Material Browser** 
- Select **View>Material Browser**



2 In the **Material Browser**, under **Materials**, expand the **Built-In** folder. Scroll down to find **Structural Steel**, right-click and select **Add Material to Model**.



3 Examine the **Material Contents** section to see how COMSOL categorizes available material information with respect to what the active physics requires.

Material Contents				
Property	Name	Value	Unit	
✓ Density	rho	7850[kg/m...	kg/...	
✓ Young's modulus	E	200e9[Pa]	Pa	
✓ Poisson's ratio	nu	0.33		
Relative permeability	mur	1	1	
Heat capacity at constant...	Cp	475[J/(kg*K)]	J/(kg...	
Thermal conductivity	k	44.5[W/(m...	W/(...	
Electrical conductivity	sigma	4.032e6[S/...	S/m	
Relative permittivity	epsilon_r	1	1	
Coefficient of thermal ex...	alpha	12.3e-6[1/K]	1/K	



Also see the busbar tutorial sections “Materials” on page 17 and “Customizing Materials” on page 41 to learn more about working with the Material Browser.

GLOBAL DEFINITIONS

Parameters

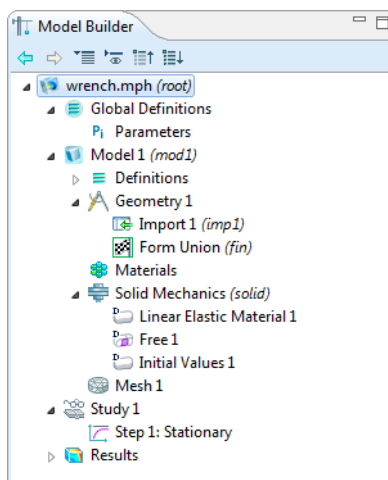
- 1 In the **Model Builder**, right-click **Global Definitions** and choose **Parameters** P_1 .
- 2 Go to the **Parameters** settings window. Under **Parameters** in the **Parameters** table (or under the table in the fields), enter these settings:
 - In the **Name** column or field, enter **F**.
 - In the **Expression** column or field, enter **150[N]**. The **Value** column is automatically updated based on the expression entered.
 - In the **Description** column or field, enter **Applied force**.

2

Parameters			
Name	Expression	Value	Description
F	150[N]	150.00 N	Applied force

In the busbar tutorial, the section “Global Definitions” on page 12 and “Parameters, Functions, Variables, and Model Couplings” on page 36 shows you more about working with parameters.

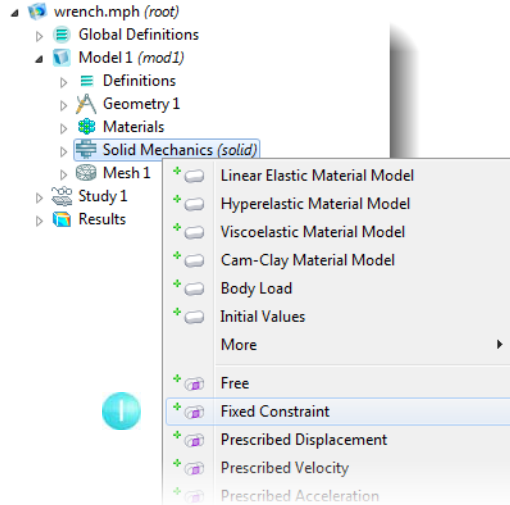
So far, you have added the physics and study, imported a geometry, added the material and defined one parameter. The **Model Builder** node sequence should match the figure. The default feature nodes under **Solid Mechanics** are indicated by a ‘D’ in the upper left corner of the node.



DOMAIN PHYSICS AND BOUNDARY CONDITIONS

With the geometry and materials defined, you are now ready to revisit the governing physics introduced in the Model Wizard section.

- 1 In the **Model Builder**, right-click **Solid Mechanics (solid)** and at the boundary level, select **Fixed Constraint**.

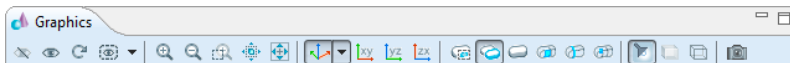
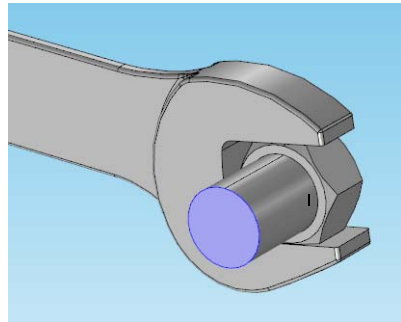


- 2 In the **Graphics** window, rotate the geometry by left-clicking and dragging into the position shown. Then left-click the cut-face of the partially modeled bolt (which turns the boundary red) and right-click to select it (which turns the boundary blue).

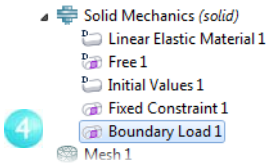
Also see “Working with 3D Geometry in the Graphics Window” on page 105.

- 3 Click the **Go to Default 3D View** button on the **Graphics** toolbar to restore the geometry to the default view.

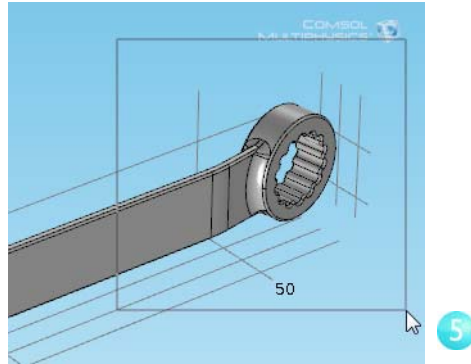
- 2 Cross-check: Boundary 35.



- 4 In the **Model Builder**, right-click **Solid Mechanics (solid)** and at the boundary level, select **Boundary Load**. A **Boundary Load** node is added to the **Model Builder** sequence.

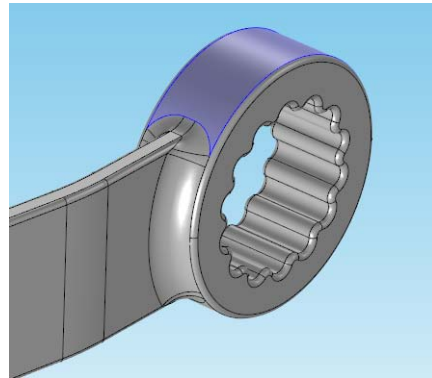


- 5 In the **Graphics** window, click the **Zoom Box** button on the toolbar and drag the mouse to highlight the area shown in the figure to the right. Release the mouse button.

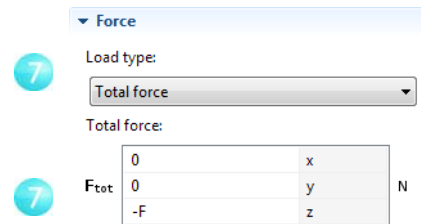


- 6 Select the top socket face (Boundary 111) by left-clicking to highlight the boundary in red and right-clicking it to highlight it in blue and add it to the **Selection** list.

- 6 Cross-check: Boundary 111.



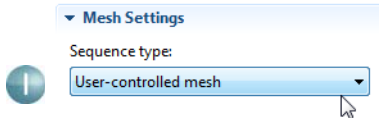
- 7 In the **Boundary Load** settings window, under **Force**, select **Total force** as the **Load type** and enter $-F$. The negative sign indicates the negative z direction (downward).



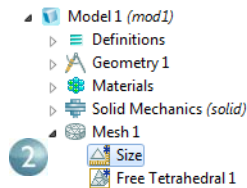
MESH

Because the geometry contains small edges and faces, reduce the size of the minimum element.


- 1 In the **Model Builder**, under **Model 1** click **Mesh 1** . In the **Mesh** settings window, under **Mesh** settings, from the **Sequence type** list select **User-controlled mesh**.



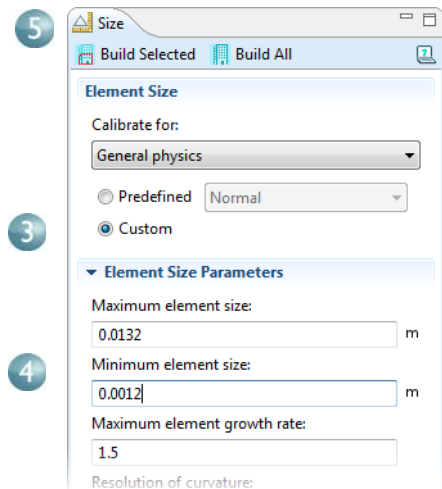
- 2 Under **Mesh 1** click the **Size**  node.



Note: The asterisks in the upper-right corner of the mesh nodes are dynamic feedback icons indicating that these nodes are being edited.










- 3 In the **Size** settings window under **Element Size**, click the **Custom** button.
- 4 Under **Element Size Parameters**, in the **Minimum element size** field, enter 0.0012.
- 5 Click the **Build All** button  on the **Size** settings window toolbar.

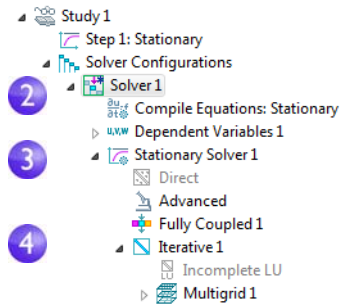
After a few seconds, the mesh builds. Zoom in to the mesh and have a look at the distribution.



STUDY

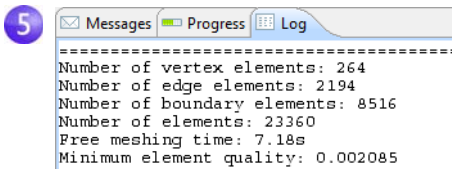
The following steps are needed to set up an iterative solver. By using such a solver, you can significantly reduce the memory amount needed for the calculations. If your computer has more than 2 GB of memory, you can skip these steps and go directly to step 6 to compute the solution.

- 1 In the **Model Builder**, right-click **Study 1**  and choose **Show Default Solver** .
- 2 Expand the **Study 1>Solver Configurations>Solver 1**  node.
- 3 Right-click **Stationary Solver 1**  and choose **Iterative** .
- 4 Right-click **Iterative 1**  and choose **Multigrid** .
- 5 Right-click **Study 1**  and select **Compute**  (or press F8).



After a few seconds of computation time, the default plot is displayed in the **Graphics** window.

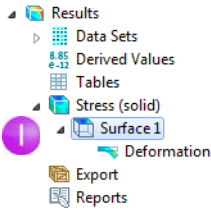
You can find other useful information about the computation by clicking the **Messages** and **Log** tabs under the **Graphics** window to see the kind of information available to you. These windows can also be opened from the main **View** menu.



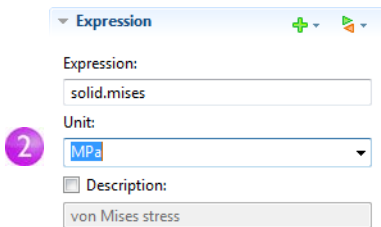
DISPLAYING RESULTS

The default von Mises stress is displayed in the Graphics window in a Surface plot with the displacement visualized using a Deformation subnode. Change the default unit (N/m^2) to the more suitable MPa.

1 In the **Model Builder**, expand the **Results>Stress (solid)** node, then click **Surface 1**.

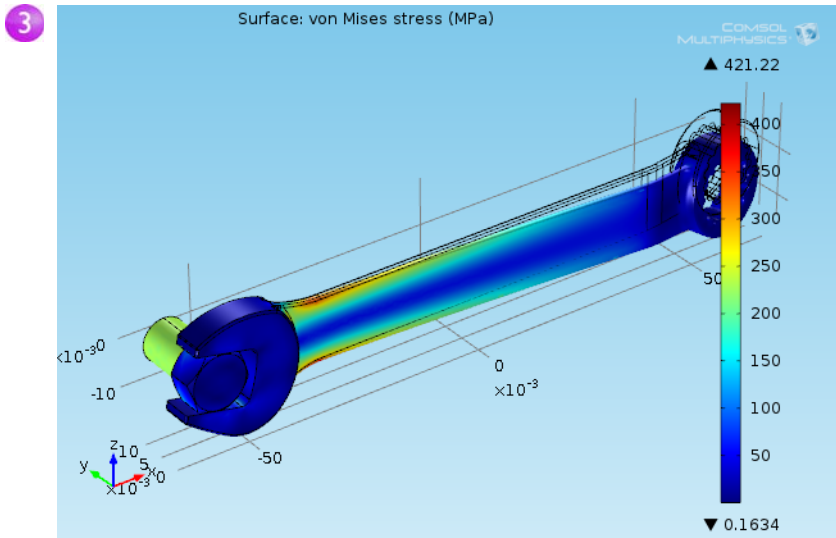


2 In the settings window under **Expression**, from the **Unit** list select **MPa** (or enter MPa in the field).







3 Click the **Plot** button and the **Zoom Extents** button on the **Graphics** toolbar.

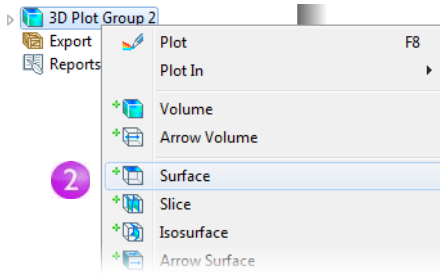
The plot is regenerated with the updated unit and shows the von Mises stress distribution in the bolt and wrench under an applied vertical load.




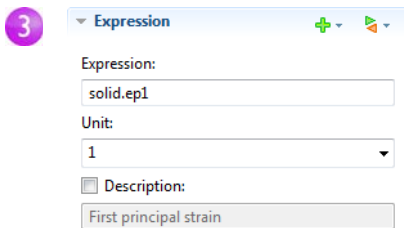
Now determine the location of greatest tension by plotting the principal strain field.


PLOTTING THE PRINCIPAL STRAIN FIELD

- 1 Right-click the **Results** node  and add a **3D Plot Group** .
- 2 Right-click the **3D Plot Group 2** node  and select **Surface** .

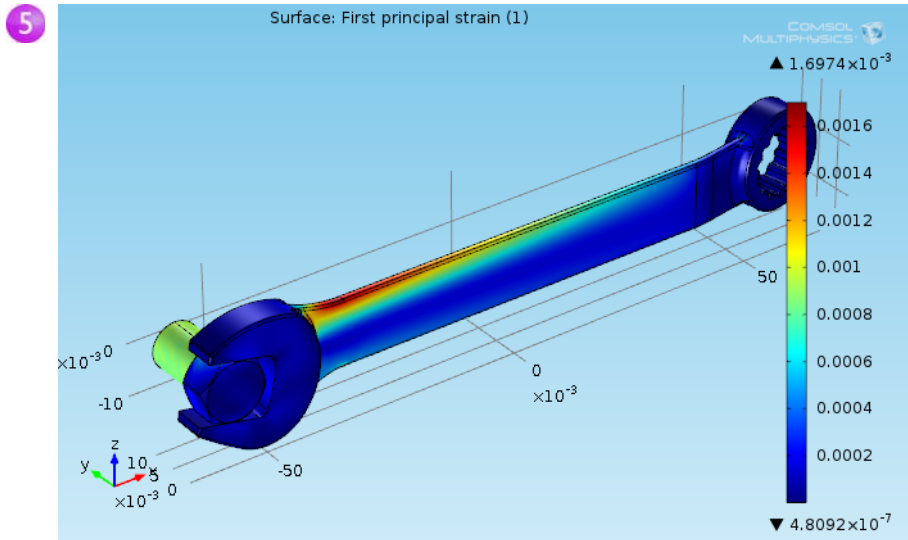


- 3 In the **Surface** settings window click the **Replace Expression** button  and select **Solid Mechanics>Strain>Principal strains> First principal strain (solid.ep1)**. When you know the variable name, as in this exercise, you can also enter `solid.ep1` in the **Expression** field.



- 4 Click the **Plot** button .
- 5 In the **Model Builder**, click **3D Plot Group 2**. Press F2 and in the **Rename 3D Plot Group** window, enter **First Principal Strain**. Click **OK**.

Notice that the maximum principal strain is lower than 2%, a result that satisfies the small strain assumption.




This concludes the wrench tutorial. The following sections provide you with some additional concepts to get you started with modeling in COMSOL.

COMSOL Multiphysics—Additional Concepts

The next sections provide some additional concepts about working in COMSOL Multiphysics, including an explanation about geometric entities and what node types are shown to help you visualize your modeling. A introduction to the Model Builder and some of the terminology and navigation tools is included in the key to the table, and the last section describes the COMSOL Multiphysics file types.
























About Geometric Entities and the Different Node Types









The Model Builder has many types of nodes to help you visualize the model structure. After you click the **Finish** button () on the **Model Wizard** window, a **Model** node is added to the Model Builder based on the space dimension selected in the first step.

The Model node has different images for each node's space dimension. The nodes under the physics also indicate several things—the space dimension of the geometric entity applied to the node (at the domain, boundary, edge, or point level), and other useful pieces of information, such as whether it is a *default* node or a *pair*. The geometric entity name varies based on space dimension, but the nodes clearly indicate the type of node you are working with. Also see “Working with 3D Geometry in the Graphics Window” for some more information about geometric entities and how to move and rotate geometry in the Graphics window.

GEOMETRIC ENTITY NAMES IN DIFFERENT SPACE DIMENSIONS

ENTITY DIMENSION	NAME IN 3D	NAME IN 2D	NAME IN 1D	NAME IN 0D
3D	domain	—	—	—
2D	boundary	domain	—	—
1D	edge	boundary	domain	—
0D	vertex	vertex	boundary	domain

SPACE DIMENSION	DOMAIN	BOUNDARY	EDGE	POINT	DEFAULT NODE EXAMPLE	PAIR
 3D						
 2D			—			
 2D axisymmetric			—			
 1D	—		—	—		

SPACE DIMENSION	DOMAIN	BOUNDARY	EDGE	POINT	DEFAULT NODE EXAMPLE	PAIR
 ID axisymmetric						
 0D	not applicable					



Additional node types include a whether it is an *exclusive* node, for example, a *contributing* node, if the node is *overridden* by another node, or if it *overrides* other nodes.

EXCLUSIVE NODES


The use of an *exclusive node* means that only one node can be active for a given selection. That is, if you add another exclusive node (for example, an identical feature node) with the same selection, the first exclusive node is overridden and thus has no effect.

Typical exclusive nodes include model equations, initial values, and boundary conditions that are constraints, such as prescribed values for displacements, temperatures, pressures, and so on, including special variants of these such as ground conditions in electromagnetics and fixed constraints in structural mechanics. Also some boundary conditions that are not constraints but have a definitive meaning are exclusive nodes—for example, electric insulation, thermal insulation, and no-flow conditions.

OVERRIDDEN AND OVERRIDING NODES







Depending on the selections for each node, an exclusive node can override another node partially. Nodes are exclusive only within their specific physics. When a node is selected in the Model Builder tree, nodes that are *overridden* by the selected node are indicated by this icon () , and nodes that *override* the selected node are indicated by this icon () .

CONTRIBUTING NODES

A *contributing node* () means you can have several of these nodes with the same selection—the software adds these together when evaluating the model. Typical contribution nodes are loads, fluxes, and source terms, where you can have more than one of each type that is active on the same domain or boundary, for example.

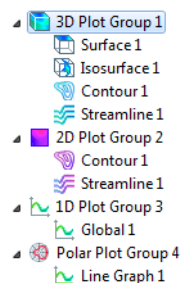
OTHER NODE TYPES

Other indicators include show the status of nodes (is the node current, being built or edited, for example) or alert you to problems (error and warning nodes).

TYPE	EXAMPLE	DESCRIPTION
Current node		A current node is used for Geometry and Meshing nodes and indicates that the feature or sequence of steps has been built. It is a green line on the left and upper edges of the node.
Current node, but not built		This node is current, but not yet built, for example, on a Geometry node. It is a brown line on the left and upper edges of the node. This example is also displaying the asterisk indicating the node is being edited.
Editing, or in process of editing, a node		An edited node has a blue asterisk (star) in the upper right corner of the node, for example, on a Mesh node. The asterisk also appears on plot nodes when the plot has not been updated to reflect changes in the data or settings (for example, after re-solving).
Warning		A warning does not prevent the completion of a task but it might affect the accuracy or other aspects of the model. When there is a yellow triangle with an explanation mark in the lower right corner of the node, a warning message displays. Other warnings typically appear in the Log window.
Error (indicator)		An error prevents the program from completing a task. For many error types, a red X is added to the lower right-hand corner of the node where the error is occurring. For example, on a Material node, to indicate that information is missing.
Error (node)		An error prevents the program from completing a task. This is an error subnode which contains an error message that generally provide additional information. It is a node with an X in a red circle, for example, under a Solver node.

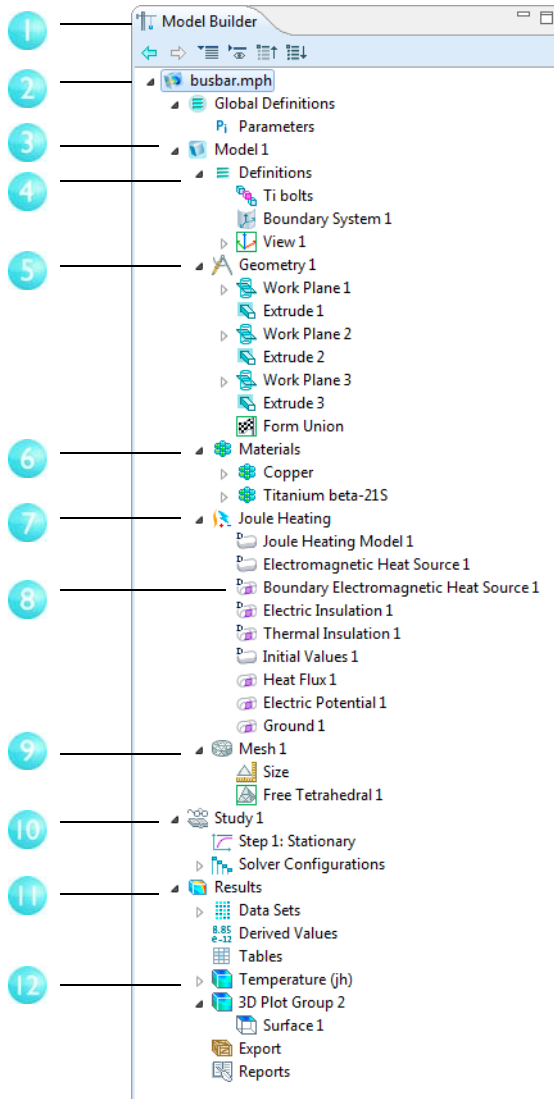
PLOT AND PLOT GROUP NODES

The plot groups and plots also indicate the space dimension and the type of plot being used. There are too many plot types to list here, but the key is that all the 1D and polar plots selected from 1D Plot Group and Polar Plot Group nodes are green, all the 2D plots selected from the 2D Plot Group node are pink, and all the 3D plots selected from the 3D Plot Group node are blue, as shown in this figure.



An Overview of the Model Builder

This figure shows you the final node sequence you have completed for the first part of this tutorial for the basic Busbar model. Go to the table after the figure to learn more about the numbered features in the figure.



- 1 This is the window name (in this example it is the Model Builder) with the toolbar underneath. Right-click any window to detach, move, size, minimize, and maximize it. Click and hold the mouse button down to drag the window and reposition it in the COMSOL Desktop.
- 2 The model file name is the Root node. Click this node to edit the default unit system or font and see details about the model. Right-click to add additional Model or Study nodes (and open the Model Wizard). If a model thumbnail is created (as in the section “Creating Model Images from Plots” on page 34) this is where you see the image when the Root node is clicked.
- 3 The icon next to Model 1 means it is a 3D model. The icon changes based on the space dimension (see “About Geometric Entities and the Different Node Types” on page 98). Click the Model node to edit the model identifier and settings or right-click the node to Add Physics (and open the Model Wizard) and to add additional Mesh nodes.
- 4 The Definitions and Global Definitions node (under the Root node) are where you can define features with a local or global scope, respectively. In this tutorial, there are several sections where you learn about these features—“Global Definitions” on page 12, “Defining Functions” on page 36, and “Defining Model Couplings” on page 39 introduce you to some of the features.
- 5 The Geometry node contains a sequence of nodes, or steps, that in this case create the geometry displayed in the Graphics window. It is the same sequence of nodes completed in the section “Geometry Sequences” on page 73.
- 6 Click the Materials node to open the Material settings window and get an overview of the materials contained in the model. Right-click the Materials node to open the Material Browser and define the materials. Click the nodes under Materials to view, add, and edit the material properties and contents, or change the appearance of the material in the Graphics window. See “Material Properties and Libraries” on page 41 for more information about this node.

7

This is an example of the Joule Heating interface that is added to the model using the Model Wizard. Right-click the interface node to open the context menu to add additional domain, boundary, edge, point, and pair nodes to the model. In the section “Adding Physics to a Model” on page 45 you learn more about working with more than one interface.

8

The physics nodes that you see in the Model Builder can tell you many things. This node tells you three things—it is a default node at the boundary level for a 3D model. See “About Geometric Entities and the Different Node Types” on page 98 for examples in other space dimensions.

9

The Mesh node contains the sequences of mesh operations that defines the computational meshes for the model. When there is only one mesh in the model, its Mesh node appears directly under the Model node. In the section “Mesh Sequences” on page 43 you learn more about this feature.

For this, and all nodes, right-click to rename a node (or press F2) and view the node Properties, for example.

10

When you select any of the predefined study types in the Model Wizard, a Study node is added to the Model Builder as well as the corresponding study step or steps.

Most study steps are used to control the form of the equations, which physics are included and which meshes are used. These study steps correspond to a part of a solver sequence, which contains nodes that define variables to solve for, the solvers and their settings, and additional sequence nodes for storing the solution, for example. The section “Running a Study Sequence—Fluid Flow and Joule Heating” on page 62 shows you the power of these features.

Right-click the Study node to Compute the solution and add additional Study Steps, as is done in the section “Study” on page 28, for example.



The Results node has all the features you need during results analysis and visualization. These main operation features are available: data sets, plots and plot groups, derived values, tables, the ability to export data and images, and to generate reports. Right-click any of these nodes to open the corresponding context menu, as shown in the section “Results” on page 29.



Click the plot nodes under Results to view the final result in the Graphics window. Like the nodes under the physics, both the Plot Group and individual plot icons indicate the space dimension, as well as what plot type is being used, as described in “About Geometric Entities and the Different Node Types” on page 98. Right-click the Plot Groups to add additional plots or images to reports.

COMSOL File Formats

When you are ready to save your model, there are three COMSOL Multiphysics model file formats to choose from—MPH-files, Model Java-files, and Model M-files.

COMSOL MPH-Files are the default standard file with the extension *.mph*. The file contains both binary and text data. The mesh and solution data are stored as binary data, while all other information is stored as plain text. You can quickly save and load MPH-files. All the models in the COMSOL Multiphysics Model Library and the model libraries in the add-on modules are MPH-files. See “Opening the Model Library” for more information.

Model Java-Files are editable script files, with the extension *.java*, that contain sequences of COMSOL Multiphysics commands as Java code. You can compile these Java files and run them as separate applications. Edit the files in a text editor to add additional commands.

Model M-files are editable script files, similar to the Model Java-files, for use with MATLAB. Model M-files, which have the extension *.m*, contain a sequence of COMSOL Multiphysics commands as a MATLAB M-file. Run the Model M-files in MATLAB like any other M-file scripts. Also edit the files in a text editor to include additional COMSOL Multiphysics commands or general MATLAB commands. Running files in the Model M-file format requires the COMSOL LiveLink™ for MATLAB®.

Also see “Supported External File Formats” for more information about all the other formats supported by COMSOL.

Working with 3D Geometry in the Graphics Window

There are toolbar buttons, mouse click options, window settings, and keyboard shortcuts available to help move, select, and highlight geometric entities. Often there is more than one way to do the same thing. In COMSOL, every level of geometry can be treated individually.

COMSOL Multiphysics highlights geometric entities at different stages of selection. Selected lines also have a thicker line width and selected points are larger. Each geometric entity is highlighted in a sequence—red, blue, green, or with no highlight—to indicate its status. It cycles between red (left-click to select the geometric entity), blue (right-click to add it to the selection), green (left-click to show that it is added), and no highlight (right-click to cancel the selection). The table below lists the many ways to move and rotate the geometry.

TASK	ACTION AND RESULT	OPERATION ORDER
Rotate the geometry about the axes:	Left-click and hold down the mouse button while dragging it in any direction. This rotates the scene around the axes parallel to the screen X- and Y-axes with origin in the scene rotation point.	left-click
Move the visible frame on the image plane in any direction, (like using a camera shift lens):	Right-click and hold down the mouse button while dragging it in any direction.	right-click
Rotate about the X- and Y axes in the image plane (tilt and pan the camera):	Press Ctrl and left-click. While holding down both buttons, drag the mouse in any direction. This places the rotation coordinate system in the camera and rotates around the axes parallel to the screen X- and Y-axes.	Ctrl+left-click
Move the camera in the plane parallel to the image plane:	Press Ctrl and right-click. While holding down both buttons, drag the mouse in any direction.	Ctrl+right-click
Rotate the camera around the axis:	Press Ctrl+Alt, then left-click. While holding down all buttons, drag the mouse in any direction.	Ctrl+Alt+ left-click
Move the scene in the plane orthogonal to the direction the camera looks at:	Right-click then press Ctrl. While holding down both buttons, drag the mouse in any direction.	right-click+Ctrl
Move the scene in the plane orthogonal to the axis between the camera and the scene rotation point:	Right-click the mouse and press Alt. While holding down both buttons, drag the mouse in any direction.	right-click+Alt

TASK	ACTION AND RESULT	OPERATION ORDER
Move the camera into and away from the object (dolly in/out):	Click the middle mouse button and then press Ctrl. While holding down both buttons, drag the mouse in any direction.	middle-click+Ctrl
Rotate the camera about its axis between the camera and the scene rotation point (roll direction):	Press Alt, then left-click. While holding down both buttons, drag the mouse in any direction.	Alt+left-click
Move the camera along the axis between the camera and the scene rotation point:	Press Alt, then middle-click. While holding down both buttons, drag the mouse in any direction.	Alt+middle-click

Keyboard Shortcuts

SHORTCUT (WINDOWS, LINUX)	SHORTCUT (MAC)	ACTION
F1	F1	Display dynamic help for the selected node
F2	F2	Rename the selected node, file, or folder
F3	F3	Disable selected nodes
F4	F4	Enable selected nodes
F5	F5	Update solution
F7	F7	Build the selected node in the geometry and mesh branches, or compute to the selected node in the solver sequence.
F8	F8	Build the geometry, build the mesh, compute the entire solver sequence, update results data, or update the plot
Del	Del	Delete selected nodes
Left arrow (Windows); Shift + left arrow (Linux)	Left arrow	Expand a branch in the Model Builder
Right arrow (Windows); Shift + right arrow (Linux)	Right arrow	Collapse a branch in the Model Builder
Up arrow	Up arrow	Move to the node above in the Model Builder
Down arrow	Down arrow	Move to the node below in the Model Builder
Alt+left arrow	Ctrl+left arrow	Move to the previously selected node in the Model Builder
Alt+right arrow	Ctrl+right arrow	Move to the next selected node in the Model Builder
Ctrl+A	Command+A	Select all domains, boundaries, edges, or points; select all cells in a table
Ctrl+D	Command+D	Deselect all domains, boundaries, edges, or points
Ctrl+C	Command+C	Copy text in fields
Ctrl+V	Command+V	Paste copied text
Ctrl+Z	Command+Z	Undo the last operation
Ctrl+Y	Ctrl+Shift+Z	Redo the last undone operation
Ctrl+up arrow	Command+up arrow	Move a definitions node, geometry node, physics node (except default nodes), material node, mesh node, study step node, or results node up one step

SHORTCUT (WINDOWS, LINUX)	SHORTCUT (MAC)	ACTION
Ctrl+down arrow	Command+down arrow	Move a definitions node, geometry node, physics node (except default nodes), material node, mesh node, study step node, or results node down one step
Ctrl+Tab	Ctrl+Tab	Switch focus to the next window on the desktop
Ctrl+Shift+Tab	Ctrl+Shift+Tab	Switch focus to the previous window on the desktop
Ctrl+Alt+left arrow	Command+Alt+left arrow	Switch focus to the Model Builder window
Ctrl+Alt+right arrow	Command+Alt+right arrow	Switch focus to the settings window
Ctrl+Alt+up arrow	Command+Alt+up arrow	Switch focus to the previous section in the settings window
Ctrl+Alt+down arrow	Command+Alt+down arrow	Switch focus to the next section in the settings window
Shift+F10 or (Windows only) Menu key	Ctrl+F10	Open the context menu
Ctrl+Space	Ctrl+Space	Open list of predefined quantities for insertion in Expression fields for plotting and results evaluation.

Finding More Information


In this section:

- “Searching for Help Content”
- “Opening the Model Library”

Searching for Help Content


When COMSOL Multiphysics is open, the **Dynamic Help** feature opens the **Help** window. The Help window has a short description of the feature node selected in the Model Builder. The window also has links to more information and access to all the online help documentation.

To open Dynamic Help:

- Press F1 (Windows, Mac, and Linux)
- Click the **Dynamic Help** button  on the main toolbar or from any settings window
- Right-click any node and select **Dynamic Help** from the context menu
- Select **Help>Dynamic Help** from the main menu


After opening the **Help** window, click any node in the Model Builder to access the description of the node. Then follow the links or enter a search term to find out more.

To open the full documentation set, which is searchable and available in PDF and HTML formats:


- Click the **Documentation**  button on the main toolbar, or
- Select **Help>Documentation** from the main menu

Then either enter a search term or navigate the **Contents** for a specific module in the documentation tree. Note that it can take up to a minute for the Help window to load and process the index the first time it is activated but after this it will load quickly. Also see “The Help Window” on page 100 of the *COMSOL Multiphysics User's Guide* for more details.



Opening the Model Library

To open the Model Library, select **View>Model Library** () from the main menu, and then search by model name or browse under a module folder name. Click to highlight any model of interest, and select **Open Model and PDF** to open both the model and the documentation explaining how to build the model. Alternatively, click the

Dynamic Help button () or select **Help>Documentation** in COMSOL to search by model name or browse by module.

The Model Library is updated on a regular basis by COMSOL in order to add new models and to improve existing models. Choose **View>Model Library Update** () to update the model library.

The MPH-files in the COMSOL model libraries can have two formats—Full MPH-files or Compact MPH-files.

- Full MPH-files, including all meshes and solutions. In the Model Library these models appear with the  icon. If the MPH-file's size exceeds 25MB, a tooltip with the text “Large file” and the file size appears when you position the cursor at the model's node in the Model Library tree.
- Compact MPH-files with all settings for the model but without built meshes and solution data to save space on the DVD (a few MPH-files have no solutions for other reasons). You can open these models to study the settings and to mesh and re-solve the models. It is also possible to download the full versions—with meshes and solutions—of most of these models through Model Library Update. In the Model Library these models appear with the  icon. If you position the cursor at a compact model in the Model Library window, a No solutions stored message appears. If a full MPH-file is available for download, the corresponding node's context menu includes a Model Library Update item.

Supported External File Formats

After completing a design study as you have done, you often want to use the results in some other context. For instance, you may want to output an optimal geometry in a dedicated CAD format. In fact, the broadly applicable nature of multiphysics simulation brings with it the need for interaction with many other scientific computing platforms. For this reason, COMSOL Multiphysics provides the capability to read and write an array of file formats. The list below contains several formats that are compatible with COMSOL. These formats are categorized according to the associated type of software.

MCAD

FILE FORMAT	READ	WRITE
AutoCAD® (.dwg, 3D only) ¹	Yes ⁵	Yes ⁵
Autodesk Inventor® (.ipt, .iam) ²	Yes	Yes ⁵
Creo™ Parametric (.prt, .asm) ²	Yes	Yes ⁵
Pro/ENGINEER® (.prt, .asm) ²	Yes	Yes ⁵
SolidWorks® (.sldprt, .sldasm) ²	Yes	Yes ⁵
SpaceClaim® (.scdoc) ³	Yes ⁵	Yes ⁵
DXF (.dxf, 2D only)	Yes	Yes
Parasolid® (.x_t, .xmt_txt, .x_b, .xmt_bin) ²	Yes	Yes
ACIS® (.sat, .sab) ²	Yes	No
Step (.STEP) ²	Yes	No
IGES (.IGES) ²	Yes	No
CATIA® V5 (.CATPart, .CATProduct) ⁵	Yes	No
VRML, vI (.vrmI, .vrl)	Yes	No
STL (.stl)	Yes	No

¹Requires LiveLink™ for AutoCAD®

²Requires one of the LiveLink™ products for AutoCAD®, Creo™ Parametric, Inventor®, Pro/ENGINEER®, SolidWorks®, or SpaceClaim®; or the CAD Import Module

³Requires LiveLink™ for SpaceClaim®

⁴Requires the CAD Import Module (or one of the LiveLink™ products for AutoCAD®, Creo™ Parametric, Inventor®, Pro/ENGINEER®, SolidWorks®, or SpaceClaim®) and the File Import for CATIA® V5

⁵From/To file via linked CAD package

ECAD

FILE FORMAT	READ	WRITE
NETEX-G (.asc) ¹	Yes	No
ODB++(X) (.xml) ¹	Yes	No
GDS (.gdx) ¹	Yes	No
SPICE (.cir) ¹	Yes	No
¹ Requires one of AC/DC Module, RF Module, or MEMS Module		

Material Databases

FILE FORMAT	READ	WRITE
NASA file (.dat) ¹	Yes	No
CHEMKIN (.dat) ¹	Yes	No
CAPE-OPEN ¹ (direct connection)	n/a	n/a
¹ Requires Chemical Reaction Engineering Module		

Mesh

FILE FORMAT	READ	WRITE
NASTRAN Bulk Data (.nas, .bdf, .nastran, .dat)	Yes	No
VRML, vI (.vrmI, .vrl)	Yes	No
STL (.stl)	Yes	No

Numerical Data, Images, and Movies

FILE FORMAT	READ	WRITE
Plain text (.txt)	Yes	Yes
Comma-separated values (.csv)	Yes	Yes
Copy and paste spreadsheet format	Yes	Yes
JPEG (.jpg)	Yes	Yes
PNG (.png)	Yes	Yes
BMP (.bmp)	Yes	Yes
GIF (.gif)	Yes	No
EPS (.eps) (2D graphs)	No	Yes
DEM (.dem)	Yes	No
Animated GIF (.gif)	No	Yes
Adobe® Flash (.swf)	No	Yes

FILE FORMAT	READ	WRITE
AVI (.avi) ¹	No	Yes
¹ Available for Windows only		

Programming Languages

FILE FORMAT	READ	WRITE
MATLAB®: Model M-File (.m) ¹	Yes	Yes
MATLAB®: Function (.m) ¹	Yes	No
Java: Model Java File (.java)	Yes	Yes
C: Function	Yes	No
¹ Requires LiveLink™ for MATLAB®		

Interpolation Data Formats

FILE FORMAT	READ	WRITE
Spreadsheet	Yes	Yes
Grid	Yes	No
Sectionwise	Yes	Yes

Built-in Functions

FUNCTION	ARGUMENTS AND DEFINITION	CALL
Analytic	<p>The function name is its identifier; for example an1.</p> <p>The function is a mathematical expression of its arguments.</p> <p>Example: Given the arguments x and y, its definition is sin(x)*cos(y).</p> <p>The function has arbitrary number of arguments.</p>	<p>The name of the function with comma-separated arguments within parenthesis. For example:</p> <p>an1(x,y)</p>
Elevation	<p>The function name is its identifier; for example elev1.</p> <p>Used to import geospatial elevation data from digital elevation models and map the elevation data to a function of x and y. A DEM file contains elevation data for a portion of the Earth's surface. The resulting function behaves essentially like a grid-based interpolation function.</p>	<p>The name of the function with comma-separated arguments within parenthesis. For example:</p> <p>elev1(x,y)</p>

FUNCTION	ARGUMENTS AND DEFINITION	CALL
Gaussian Pulse	<p>The function name is its identifier, for example gp1.</p> <p>The Gaussian pulse function defines a bell-shaped curve according to the expression</p> $y(s) = \frac{1}{\sigma\sqrt{2\pi}} e^{\frac{-(s-s_0)^2}{2\sigma^2}}$ <p>It is defined by the mean parameter, s_0, and the standard deviation, σ.</p> <p>The function has one argument.</p>	<p>The name of the function with a single argument within parenthesis. For example:</p> <p>gp1 (x)</p>
Image	<p>The function name is its identifier, for example im1.</p> <p>Used to import an image (in BMP, JPEG, PNG, or GIF format) and map the image's RGB data to a scalar (single channel) function output value. By default the function's output uses the mapping (R+G+B)/3.</p>	<p>The name of the function with comma-separated arguments within parenthesis. For example:</p> <p>im1 (x, y)</p>
Interpolation	<p>The function name is its identifier, for example int1.</p> <p>An interpolation function is defined by a table or file containing the values of the function in discrete points.</p> <p>The file formats are the following: spreadsheet, grid, or sectionwise.</p> <p>The function has one to three arguments.</p>	<p>The name of the function with comma-separated arguments within parenthesis. For example:</p> <p>int1 (x, y, z)</p>
Piecewise	<p>The function name is its identifier, for example pw1.</p> <p>A piecewise function is created by splicing together several functions, each defined on one interval. Define the argument, extrapolation and smoothing methods, and the functions and their intervals.</p> <p>This function has one argument with different definitions on different intervals, which must not overlap or have any holes between them.</p>	<p>The name of the function with a single argument within parenthesis. For example:</p> <p>pw1 (x)</p>

FUNCTION	ARGUMENTS AND DEFINITION	CALL
Ramp	<p>The function name is its identifier; for example rm1.</p> <p>A ramp function is a linear increase with a user-defined slope that begins at some specified time.</p> <p>The function has one argument. It can also be smoothed.</p>	<p>The name of the function with a single argument within parenthesis. For example:</p> <p>rm1(x)</p>
Random	<p>The function name is its identifier; for example rn1.</p> <p>A random function generates white noise with uniform or normal distribution and has one or more arguments to simulate white noise.</p> <p>The function has arbitrary number of arguments.</p>	<p>The name of the function with comma-separated arguments within parenthesis. For example:</p> <p>rn1(x,y)</p> <p>The arguments x and y are used as a random seeds for the random function.</p>
Rectangle	<p>The function name is its identifier; for example rect1.</p> <p>A rectangle function is 1 in an interval and 0 everywhere else.</p> <p>The function has one argument.</p>	<p>The name of the function with a single argument within parenthesis. For example:</p> <p>rect1(x)</p>
Step	<p>The function name is its identifier; for example step1.</p> <p>A step function is a sharp transition from 0 to some other value (amplitude) at some location.</p> <p>The function has one argument. It can also be smoothed.</p>	<p>The name of the function with a single argument within parenthesis. For example:</p> <p>step1(x)</p>
Triangle	<p>The function name is its identifier; for example tri1.</p> <p>A triangle function is a linear increase and linear decline within an interval and 0 everywhere else.</p> <p>The function has one argument. It can also be smoothed.</p>	<p>The name of the function with a single argument within parenthesis. For example:</p> <p>tri1(x)</p>
Waveform	<p>The function name is its identifier; for example wv1.</p> <p>A waveform function is a periodic function with one of several characteristic shapes: sawtooth, sine, square, or triangle.</p> <p>The function has one argument. It can also be smoothed.</p>	<p>The name of the function with a single argument within parenthesis. For example:</p> <p>wv1(x)</p>