

# Modeling Exercise

Define the physics for a model of a busbar using the manual approach with user-defined couplings

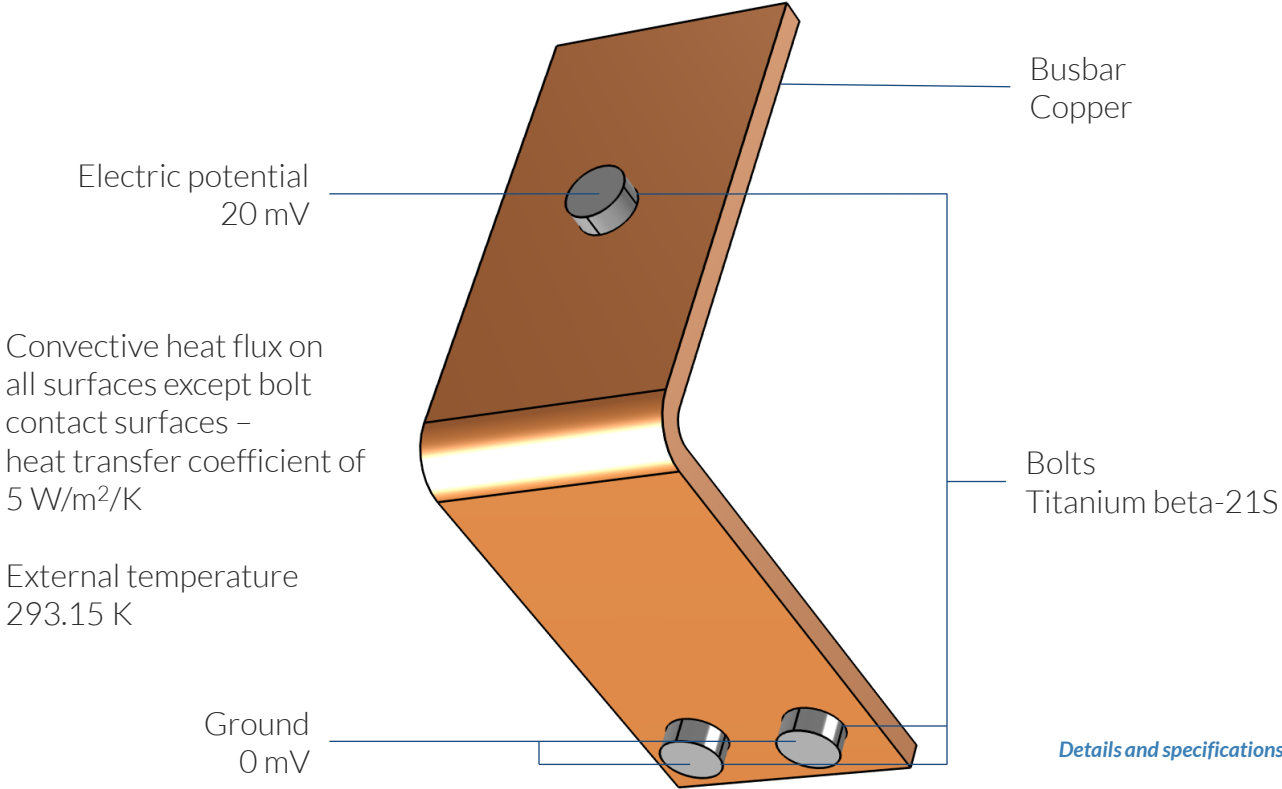
# Introduction

- This model exercise demonstrates the concept of multiphysics modeling in COMSOL Multiphysics®
- Define the physics for the model using the manual approach with user-defined couplings
  - Add and define the physics for the *Electric Currents* interface, followed by the *Heat Transfer in Solids* interface, and then manually couple the physics to simulate resistive heating using a *Heat Source* domain feature with an expression that defines the resistive losses
    - Enables you to manually implement couplings between physics interfaces for which no coupling features are available
- Important information for setting up the model can be found in the **Model Specifications** slide
  - Refer to this when building the model

# Model Overview

- A voltage difference is applied between titanium bolts at opposite ends of a copper busbar
  - This is an unwanted mode of operation of the busbar and its effect is assessed
- The voltage difference induces a current flow, causing the temperature of the busbar to rise
  - An instance of the Joule heating effect
- The busbar is cooled via natural, or free, convection
  - Modeled using a *Heat Flux* boundary condition
- Results include the electric potential and temperature distribution
  - Plot of the current density of the busbar assembly is manually generated

# Model Specifications



*Details and specifications for the busbar model setup*

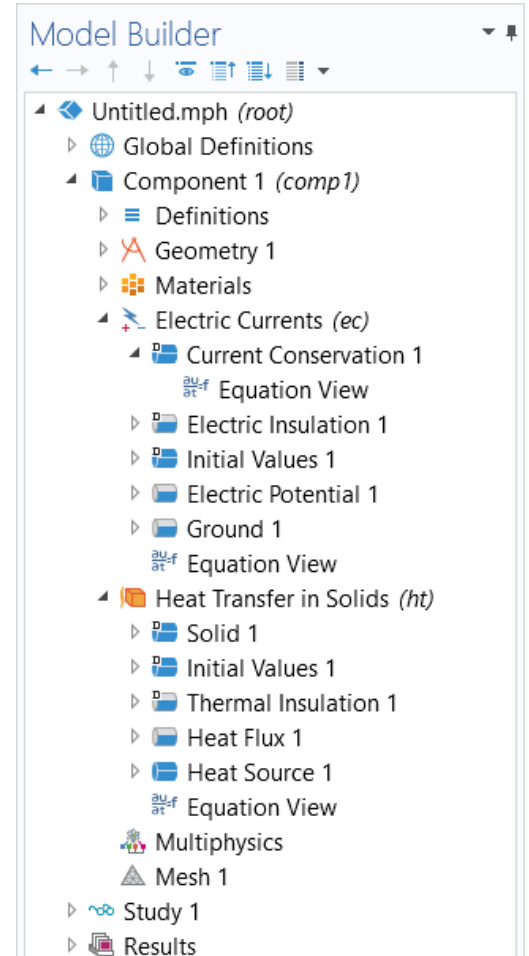
# Manual Approach with User-Defined Couplings

Define the physics for the model using user-defined multiphysics couplings

## Procedure:

1. Add the physics interface
2. Define the physics settings
3. Repeat steps 1 and 2 for each subsequent physics interface
4. Define the multiphysics couplings

*The model tree for the busbar tutorial model when the manual approach with user-defined couplings has been used*



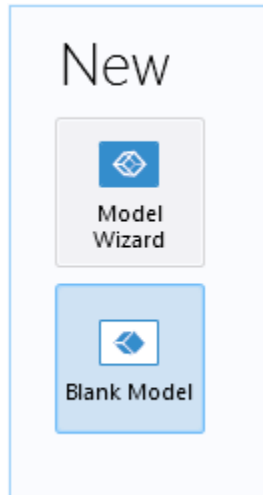
# Modeling Workflow

An outline of the steps used to set up, build, and compute this model to complete this modeling exercise is provided here:

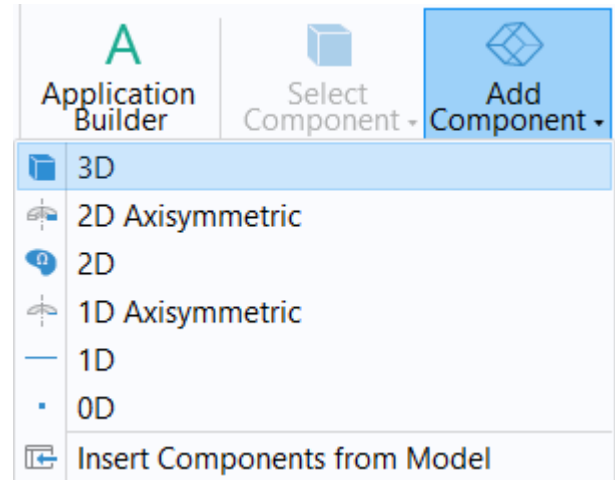
1. Set up the model
2. Import geometry
3. Assign materials
4. Define the physics
  - Add *Electric Currents* interface
  - Add *Heat Transfer in Solids* interface
  - Implement user-defined multiphysics coupling
5. Build the mesh
6. Run the study
7. Postprocess results

# Model Setup

- Open the software
- Choose a *Blank Model*
- Add a 3D model component

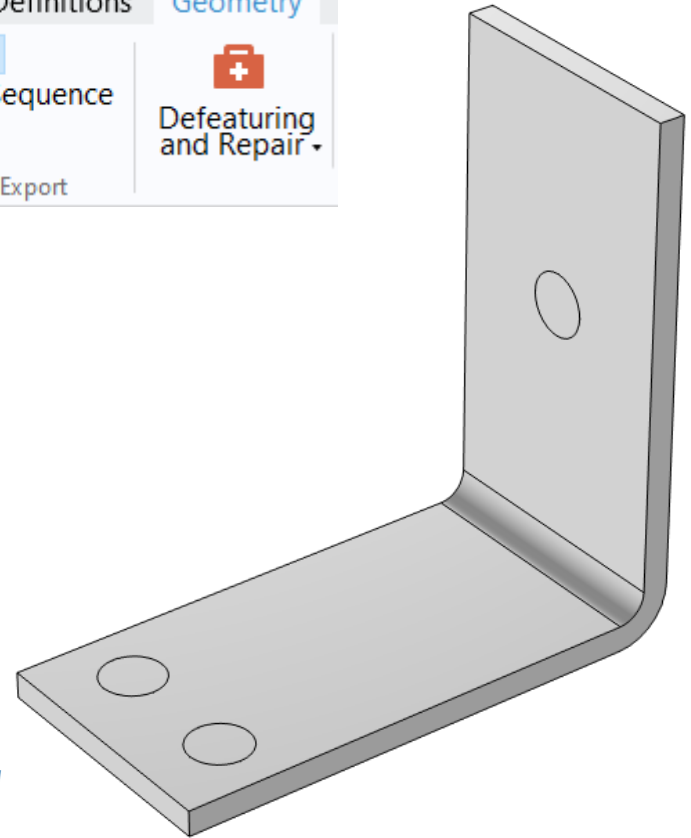
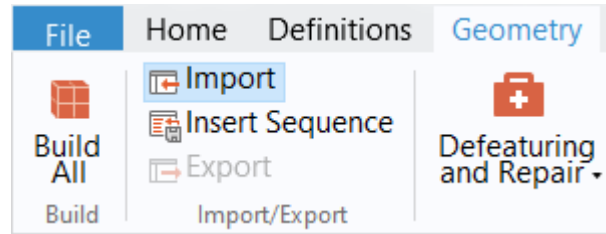


*Screenshots of the steps performed to set up the model*



# Import Geometry

- Download the geometry file *busbar.mphbin*
- Import the geometry
- Build *Form Union* operation to finalize the geometry

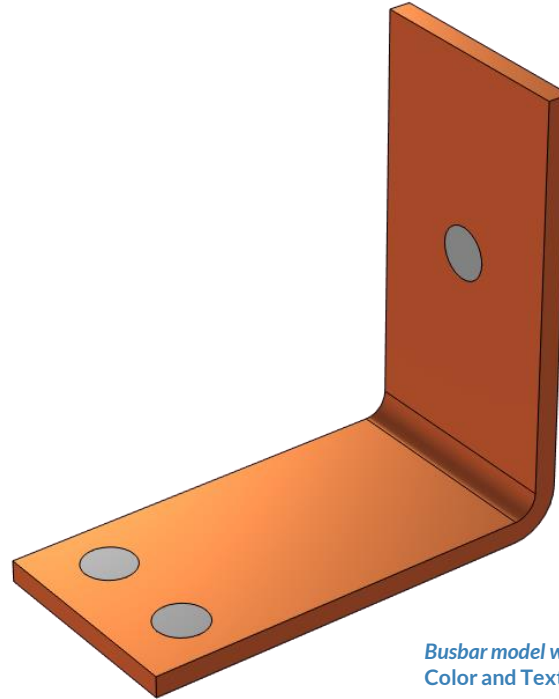


*The Import button used and the busbar model geometry*



# Assign Materials

- Busbar
  - Apply *Copper*
- Bolts
  - Apply *Titanium beta-21S*



*Busbar model with the Show Material Color and Texture option enabled*

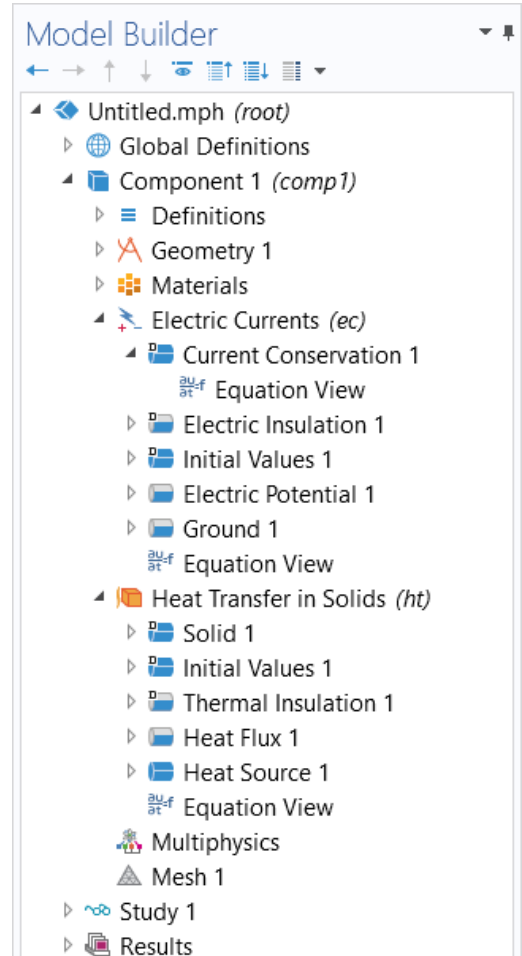
# Manual Approach with User-Defined Couplings

Define the physics for the model using user-defined multiphysics couplings

## Procedure:

1. Add the physics interfaces
  - *Electric Currents*
  - *Heat Transfer in Solids*
2. Define the physics settings
3. Repeat steps 1 and 2 for each subsequent physics interface
4. Define the multiphysics couplings
  - *Heat Source* domain feature

*The model tree for the busbar tutorial model when the manual approach with user-defined couplings has been used*

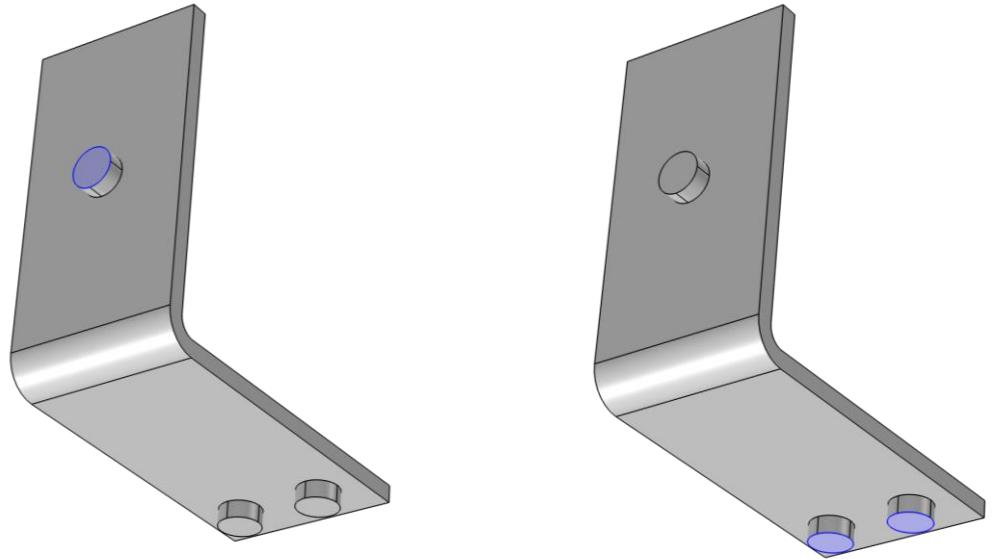


## PHYSICS SETTINGS

# Electric Currents

- Active in all domains
- Add *Electric Potential* boundary condition\*
  - Defines an electric potential on the surface
- Add *Ground* boundary condition
  - Defines zero potential on the surface

\* = Refer to model specifications for values



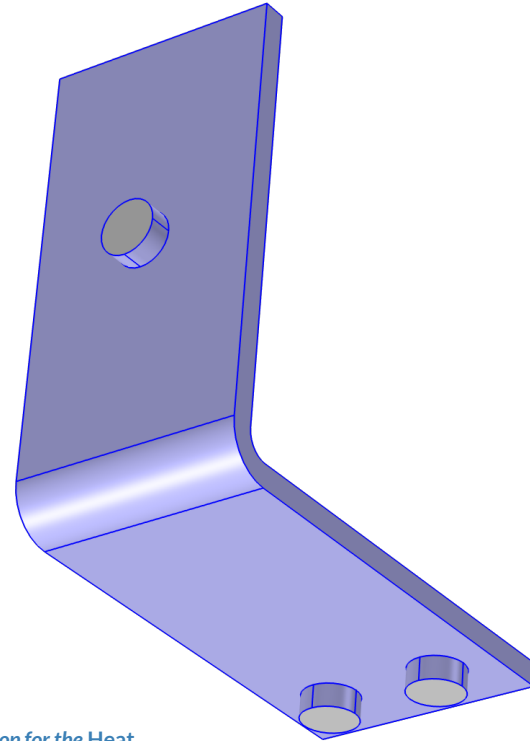
Geometry selection for the *Electric Potential* (left) and *Ground* (right) boundary conditions

## PHYSICS SETTINGS

# Heat Transfer in Solids

- Active in all domains
- Add *Heat Flux* boundary condition\*
  - Convective heat flux
  - Defines heat transfer from the device to the surrounding air, naturally occurring
- Add *Heat Source* domain feature
  - Defines heat generation within the domain
  - Used to map resistive losses as a heat source

\* = Refer to model specifications for values



*Geometry selection for the Heat Flux boundary condition*

## MULTIPHYSICS SETTINGS

# Electromagnetic Heating

- Add *Heat Source* node
  - Active in all domains
  - Choose *General source*
  - Enter expression that describes the appropriate quantity
    - Electric losses

The screenshot displays the COMSOL Model Builder interface. On the left, the 'Model Builder' tree shows a multi-physics model with nodes for Global Definitions, Component 1 (comp1), Definitions, Geometry 1, Materials, Electric Currents (ec), Heat Transfer in Solids (ht), Multiphysics, Mesh 1, Study 1, and Results. The 'Heat Source 1' node under 'Heat Transfer in Solids (ht)' is selected and highlighted in blue.

On the right, the 'Settings' window for the 'Heat Source' node is open. The 'Label' is 'Heat Source 1'. Under 'Domain Selection', the 'Selection' is set to 'All domains'. A list of domains (1-5) is shown, with domain 1 selected. The 'Material type' is set to 'Solid'. Under 'Heat Source', the 'General source' radio button is selected. The 'Q<sub>0</sub>' is set to 'User defined' with a value of '0' and units of 'W/m<sup>3</sup>'. The 'Linear source' and 'Heat rate' options are unselected.

The Settings window also displays the following equations for the other source types:

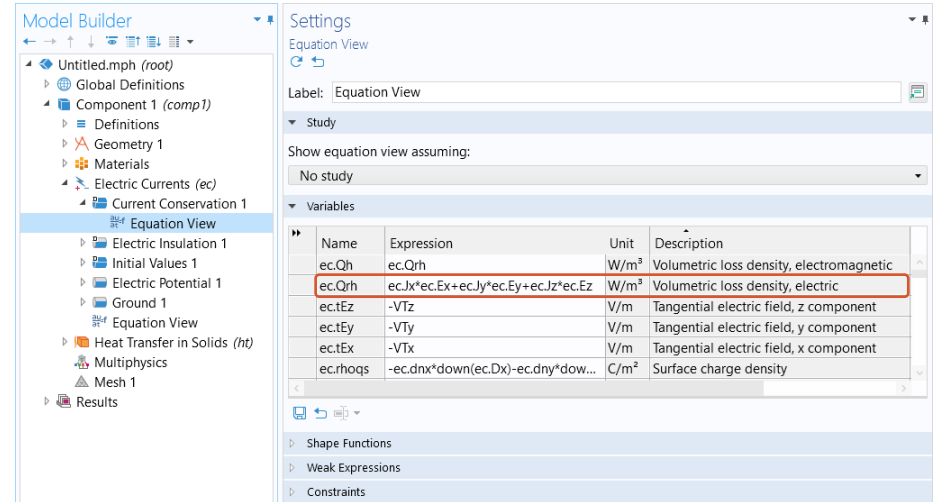
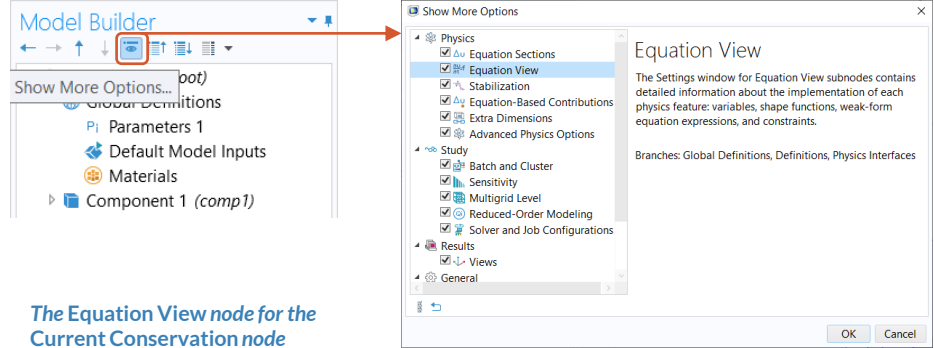
- Linear source:  $Q_0 = q_s \cdot T$
- Heat rate:  $Q_0 = \frac{P_0}{V}$

The Settings window for the Heat Source node

## MULTIPHYSICS SETTINGS

# Electromagnetic Heating

- Access predefined physics variables to formulate expression that defines the electric losses
  - Enable displaying *Equation View* nodes through the *Model Builder* toolbar
  - Select *Equation View* node under the *Current Conservation* node



## MULTIPHYSICS SETTINGS

# Electromagnetic Heating

### Define the electric losses

- Quantity is available as a predefined physics variable:

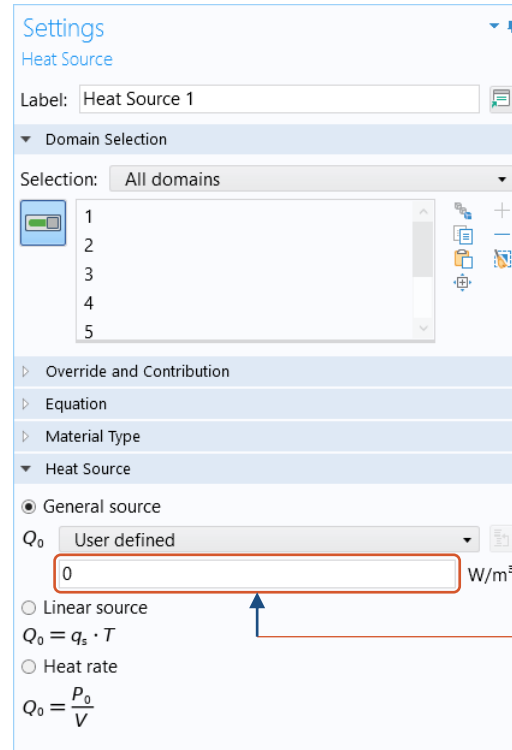
$$ec.Qrh$$

- For this model, the losses are the scalar product of the current density vector and electric field, you can enter expression:

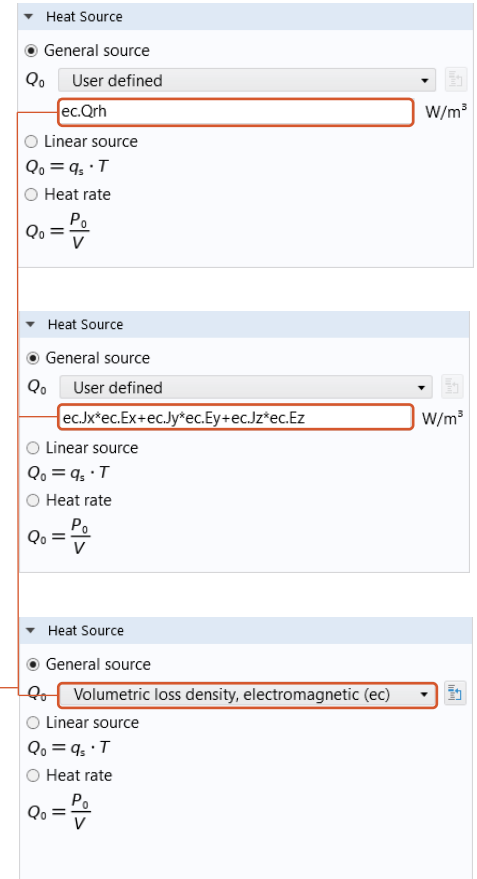
$$ec.Jx*ec.Ex+ec.Jy*ec.Ey+ec.Jz*ec.Ez$$

- Predefined heat source available that describes the resistive heating:

*Volumetric loss density, electromagnetic (ec)*



The Settings window for the Heat Source node (left) and the user-defined expressions and options that can couple the physics (right)



# Build the Mesh

Build the mesh using the default settings

Settings

Mesh

Build All

Label: Mesh 1

Mesh Settings

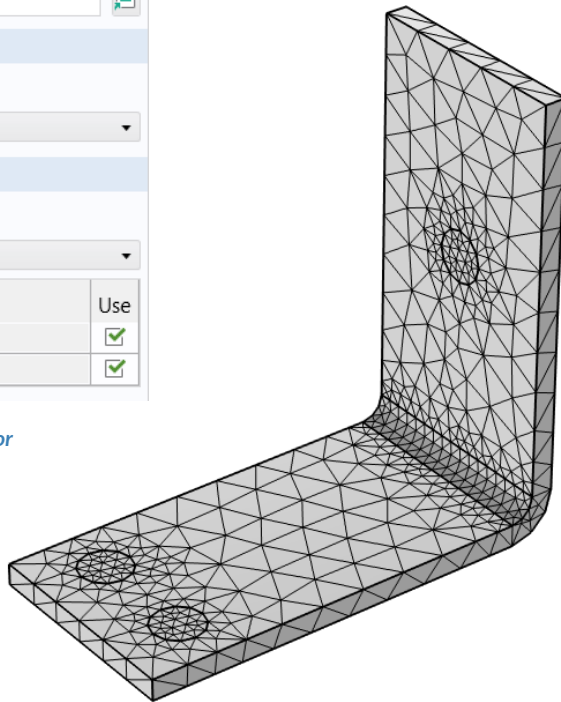
Sequence type:  
Physics-controlled mesh

Physics-Controlled Mesh

Element size:  
Normal

Contributor	Use
Electric Currents (ec)	<input checked="" type="checkbox"/>
Heat Transfer in Solids (ht)	<input checked="" type="checkbox"/>

*The settings used to generate the mesh for the busbar model, also pictured*





# Run the Study

- Add a *Stationary* study
- Compute the model

*The Add Study window, wherein the Stationary study is selected to be added to the model*

**Add Study** ▼ ↑ ×

+ Add Study

— Studies —

- ▲ General Studies
  - Stationary**
  - Time Dependent
- ▲ Preset Studies for Selected Physics Interfaces
  - ▶ Heat Transfer in Solids
  - ▶ Electric Currents
- ▶ More Studies
- ▶ Preset Studies for Some Physics Interfaces
  - Empty Study

— Physics interfaces in study —

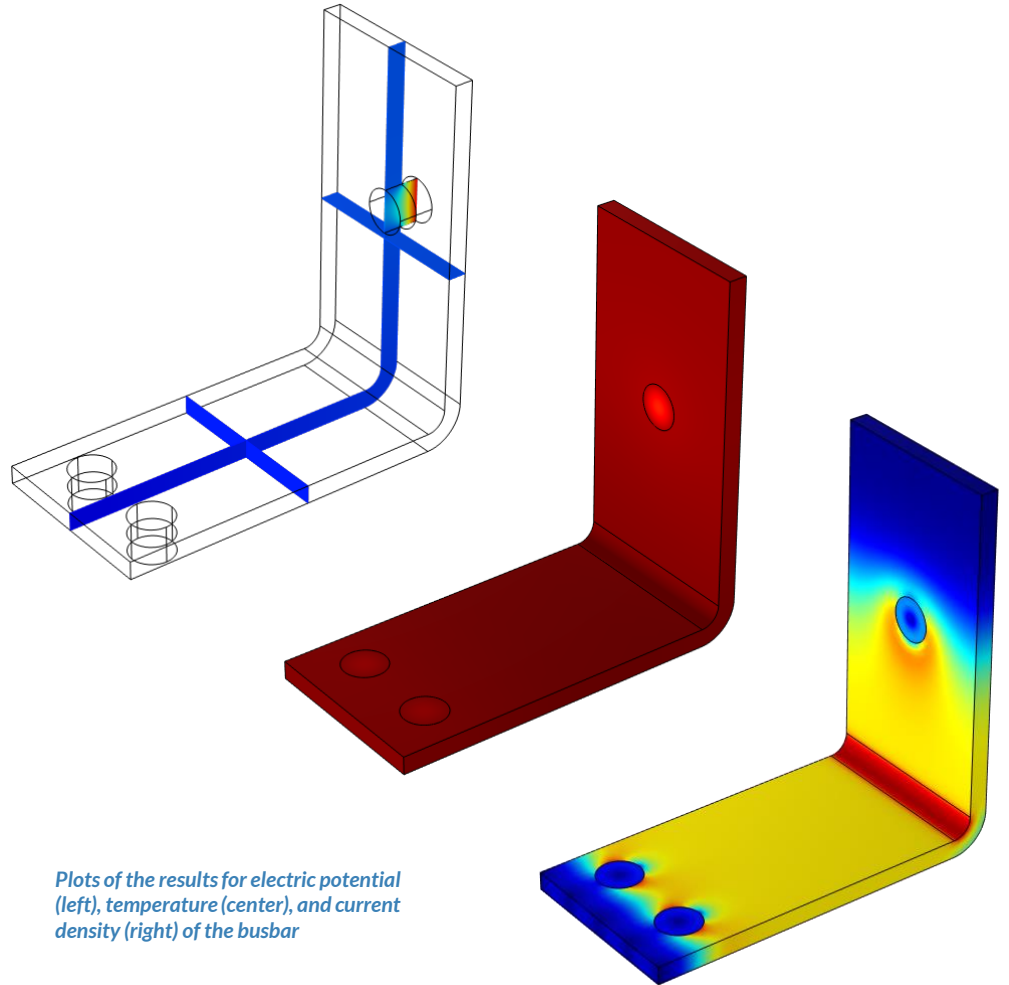
Physics	Solve
Electric Currents (ec)	<input checked="" type="checkbox"/>
Heat Transfer in Solids (ht)	<input checked="" type="checkbox"/>

— Multiphysics couplings in study —

Multiphysics couplings	Solve
------------------------	-------

# Postprocess Results

- Default plots generated by the software
  - Electric potential
  - Temperature
- Create plot for the current density
  - Add a *3D Plot Group*, rename it *Current Density*
  - Add a *Surface* plot
  - Use an expression that represents the current density norm
  - Use a *Manual Color Range*
    - *Minimum* = 0
    - *Maximum* =  $1e6$



Plots of the results for electric potential (left), temperature (center), and current density (right) of the busbar