SOLUTION

Modeling Exercise

Define the physics for a model of a busbar using the manual approach with predefined 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 predefined couplings
  - Run a single-physics simulation for the *Electric Currents* interface, followed by a multiphysics simulation including the *Heat Transfer in Solids* interface and *Electromagnetic Heating* multiphysics coupling for the resistive heating
    - Enables more quickly and easily locating and resolving any errors that may have been made in the definition of the physics phenomena involved before computing the full multiphysics model
- 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

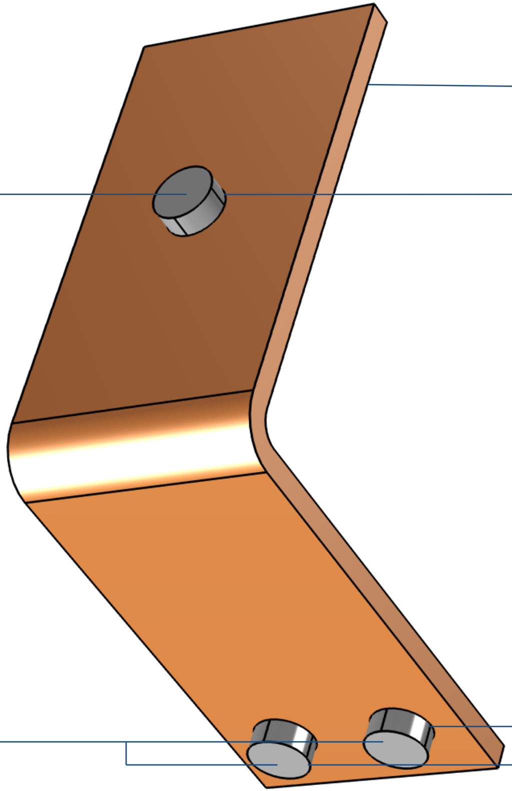
Model Specifications

Electric potential
20 mV

Convective heat flux on all surfaces except bolt contact surfaces – heat transfer coefficient of 5 $W/m^2/K$

External temperature
293.15 K

Ground
0 mV



Busbar
Copper

Bolts
Titanium beta-21S

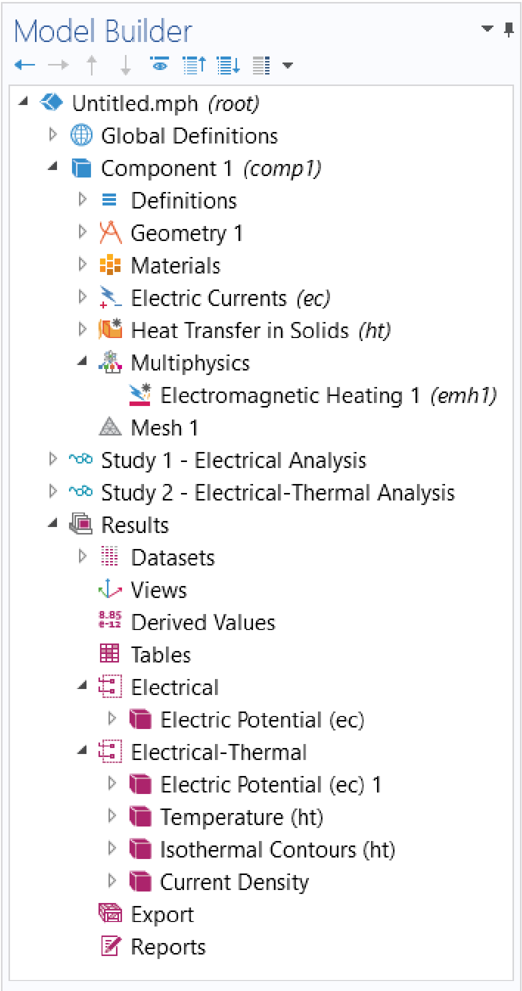
Details and specifications for the busbar model setup

Manual with Predefined Couplings Approach

Define the physics for the model using predefined multiphysics couplings

Procedure:

1. Add the physics interface
2. Define the physics settings
3. Add multiphysics couplings
   – Only applicable when multiple physics interfaces have been added
4. Compute the study
5. Check the results
6. Repeat steps 1–4 for each subsequent combination of physics



The model tree for the busbar tutorial model when the manual approach with predefined couplings has been used

Modeling Workflow

A general outline of the steps that can be used to set up, build, and compute this model to complete this modeling exercise is provided here:

Electrical Analysis

1. Set up the model
   - Add 3D model component
2. Import geometry
3. Assign materials
4. Define the physics
   - Add *Electric Currents* interface
5. Build the mesh
6. Run the study
   - Add *Stationary* study
7. Check the results

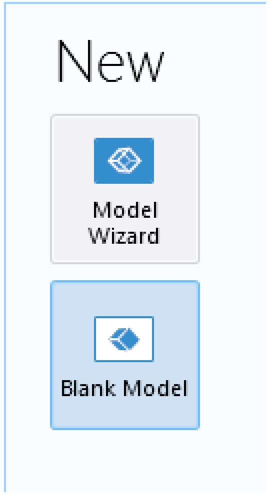
Electrical–Thermal Analysis

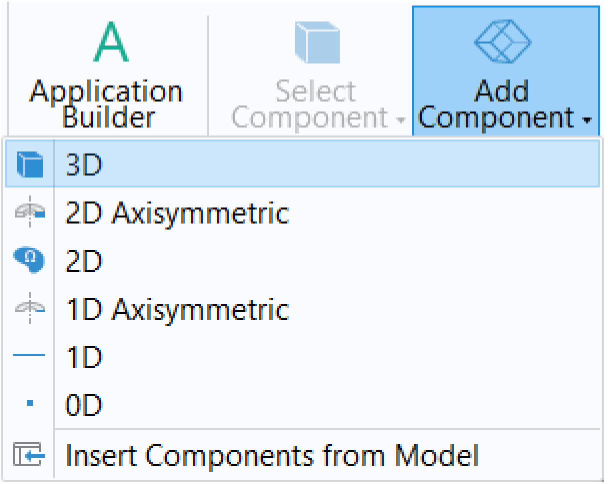
1. Define the physics
   - Add *Heat Transfer in Solids* interface
   - Add *Electromagnetic Heating* multiphysics coupling
2. Run the study
   - Add *Stationary* study
3. Check the results

* = Build the model using the procedure here and refer to the model specifications for values

Model Setup

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

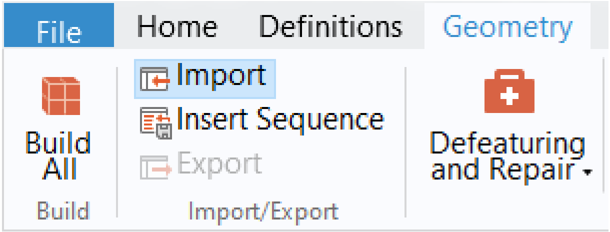


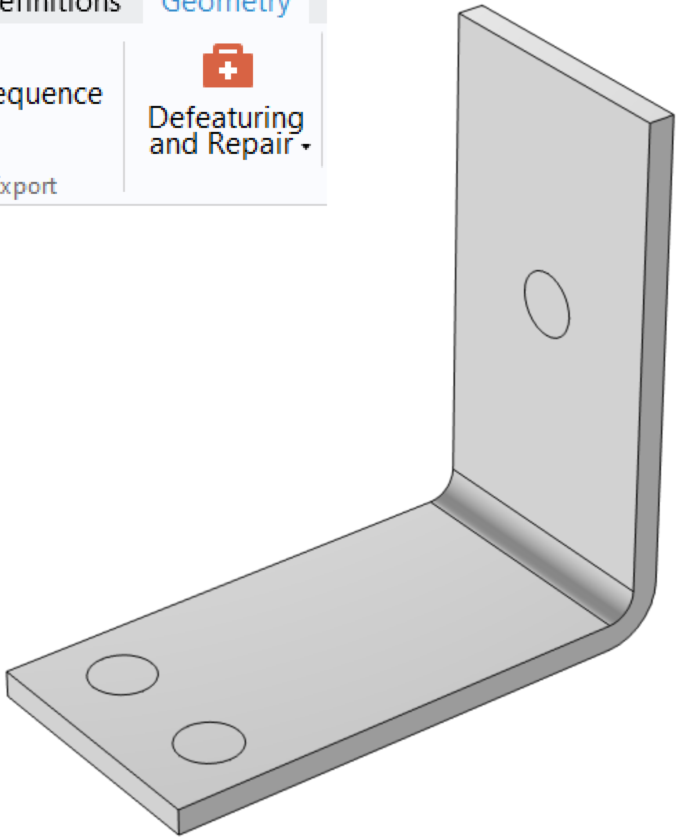


Screenshots of the steps performed to set up the model

Import Geometry

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

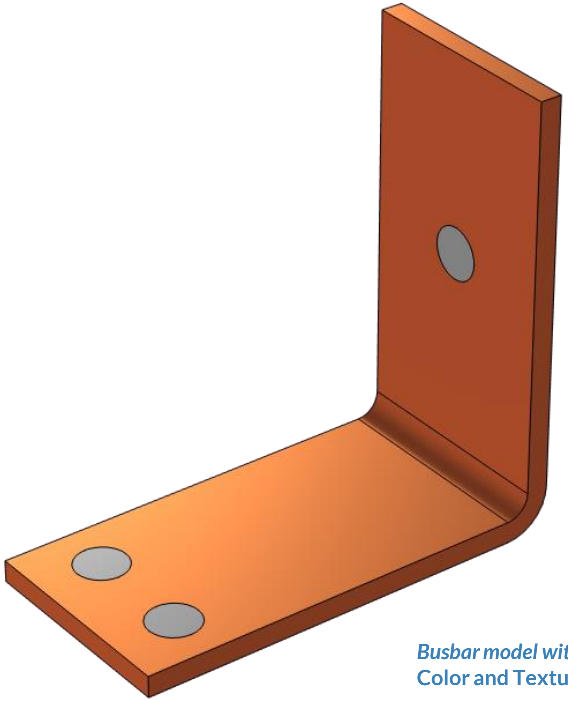




The Import button used and the busbar model geometry

Assign Materials

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



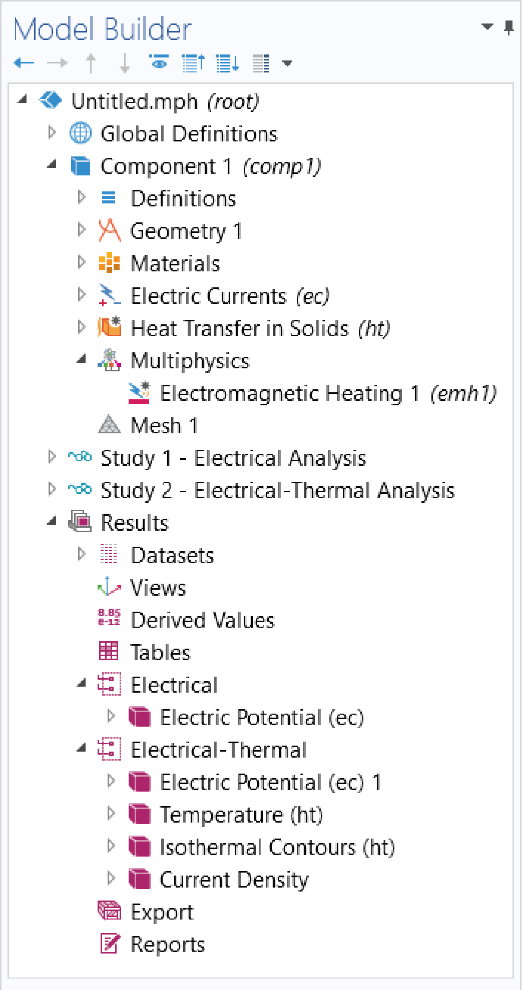
Busbar model with the Show Material Color and Texture *option enabled*

Manual with Predefined Couplings Approach

Perform two stationary studies, the first for the electric currents only and the second including heat transfer.

Procedure:

1. Electrical analysis
   - Add and define settings for the *Electric Currents* interface
2. Electrical–thermal analysis
   - Add and define settings for the *Heat Transfer in Solids* interface
   - Add the *Electromagnetic Heating* multiphysics coupling



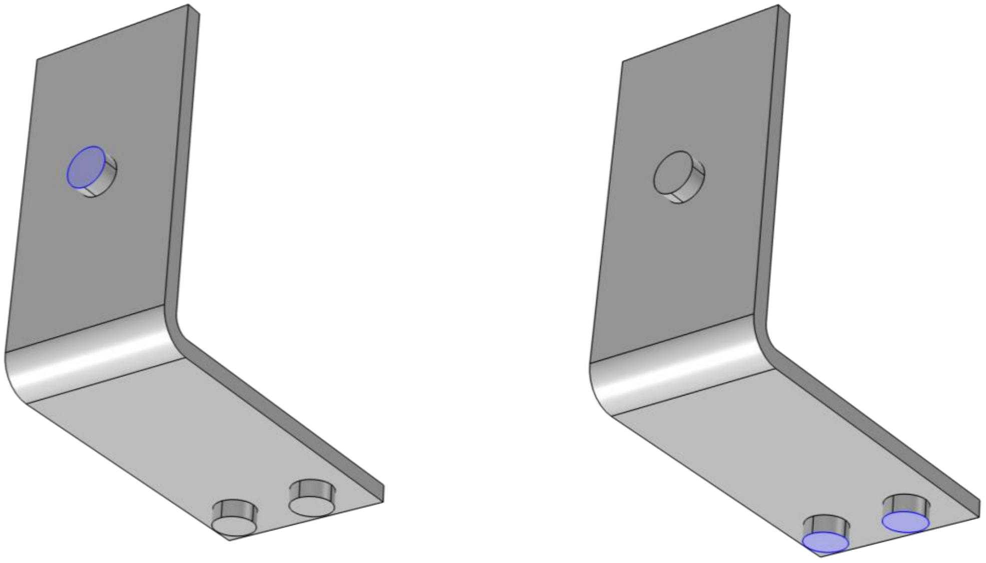
The model tree for the busbar tutorial model when the manual approach with predefined couplings has been used

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

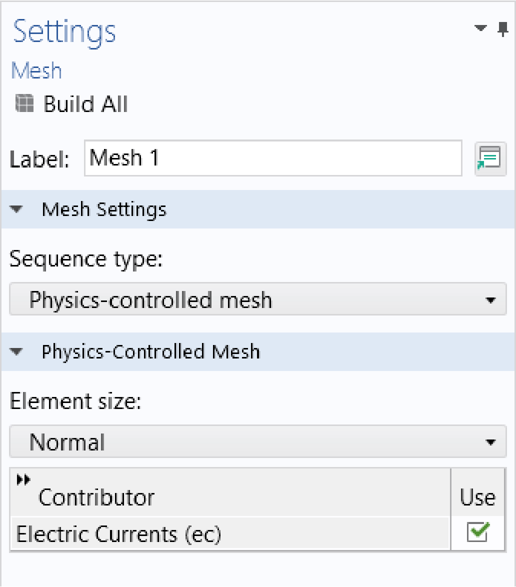
* = Refer to model specifications for values

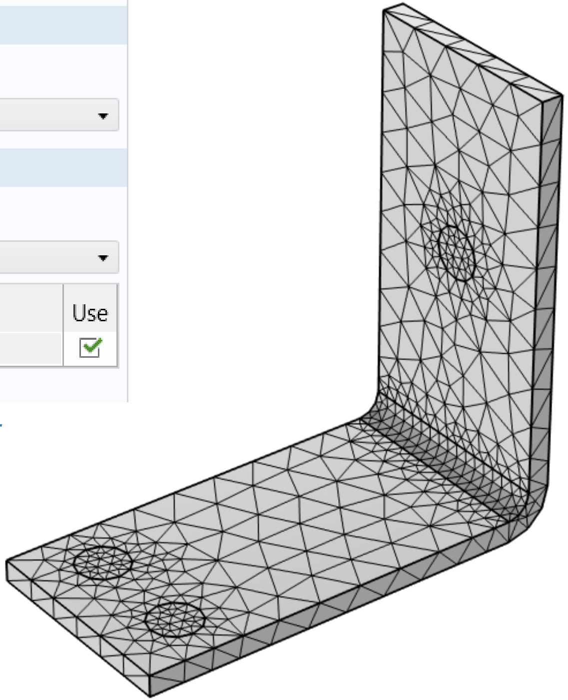


Geometry selection for the Electric Potential *(left) and* Ground *(right) boundary conditions*

Build the Mesh

Build the mesh using the default settings

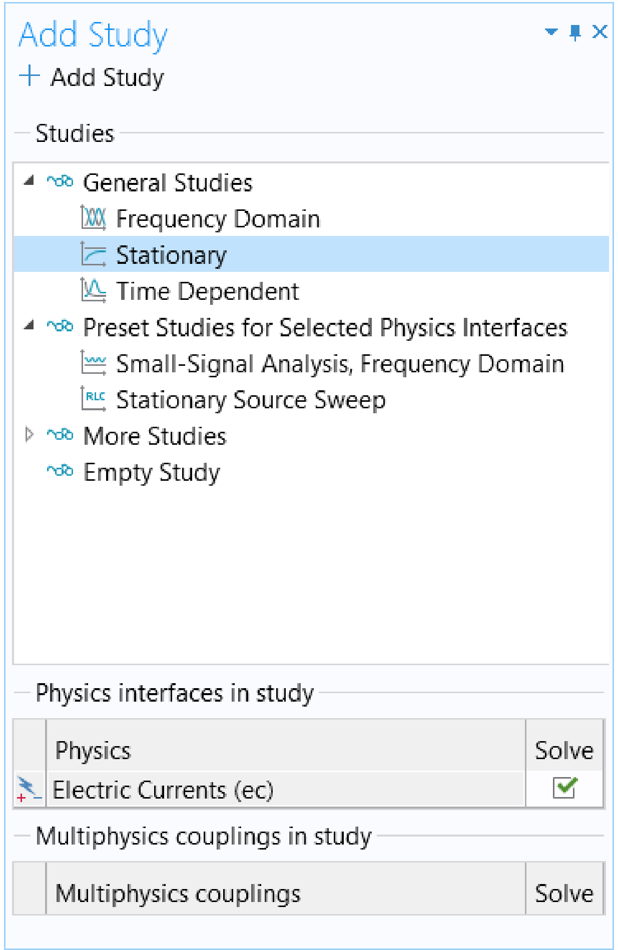




The settings used to generate the mesh for the busbar model, also pictured

Run the Study: Electrical

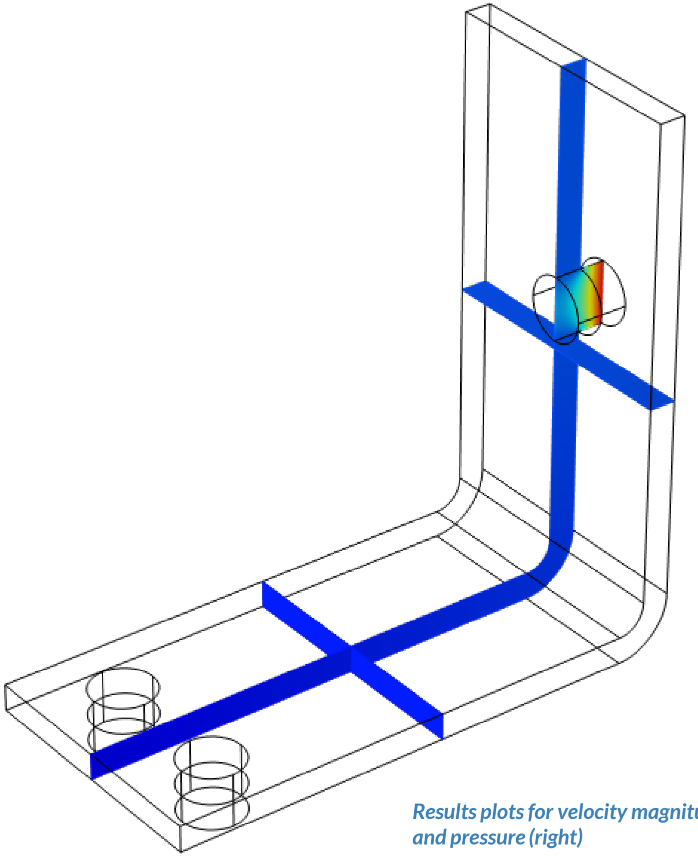
- Add a *Stationary* study
  - *Study 1*
- Change label for *Study 1* node to *Electrical Analysis*
- Compute the model



Settings for the first Stationary study added to the model

Postprocess Results: Electrical

- Default plots generated by the software
  - Electric potential

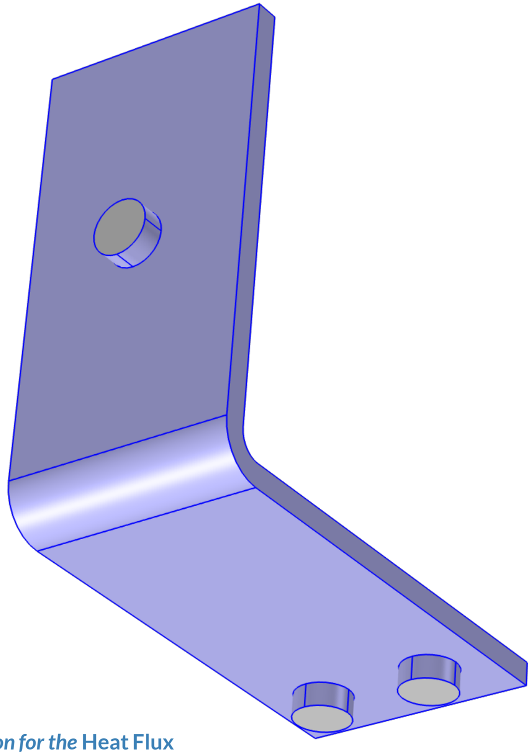


Results plots for velocity magnitude (left) and pressure (right)

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



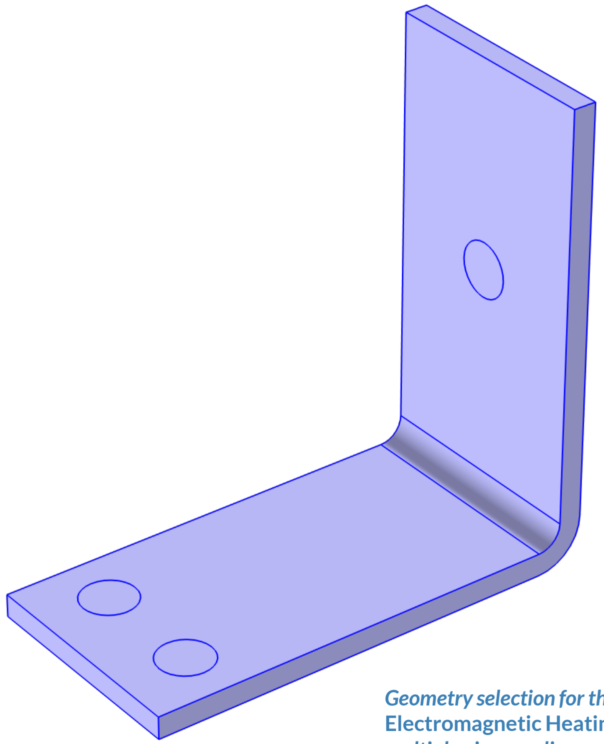
Geometry selection for the Heat Flux *boundary condition*

* = Refer to model specifications for values

MULTIPHYSICS SETTINGS

Electromagnetic Heating

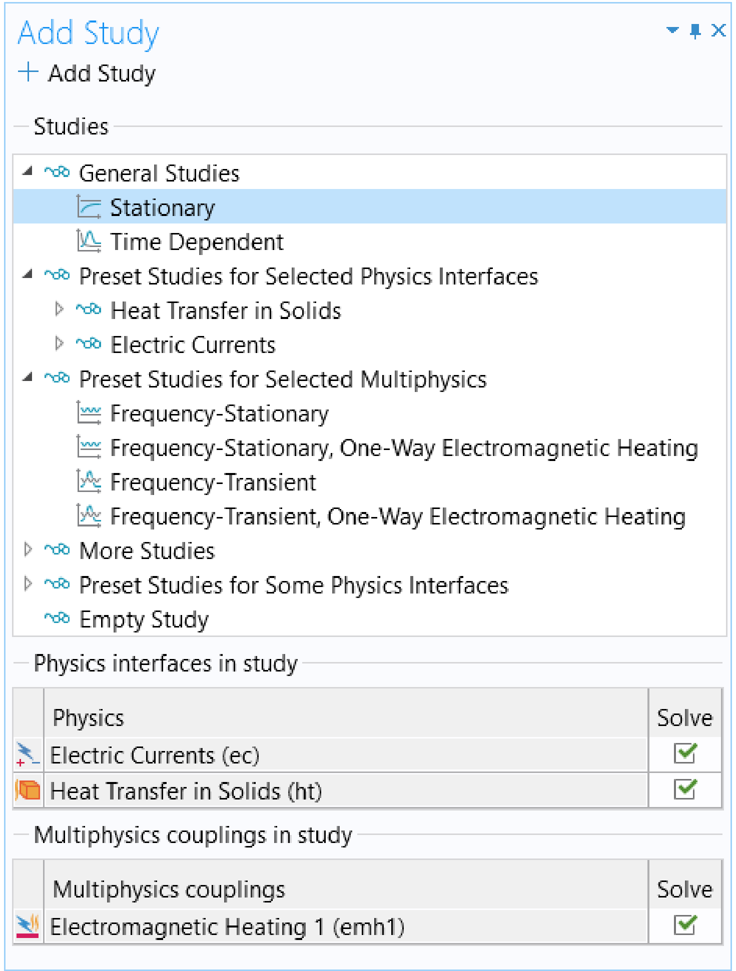
- Active in all domains
- Couples the *Electric Currents* and *Heat Transfer in Solids* physics interfaces
  - *Electric Currents*
    - Computes losses from passing electric current through the busbar
  - *Heat Transfer in Solids*
    - Incorporates resistive losses as a source of heat



Geometry selection for the Electromagnetic Heating *multiphysics coupling node*

Run the Study: Electrical–Thermal

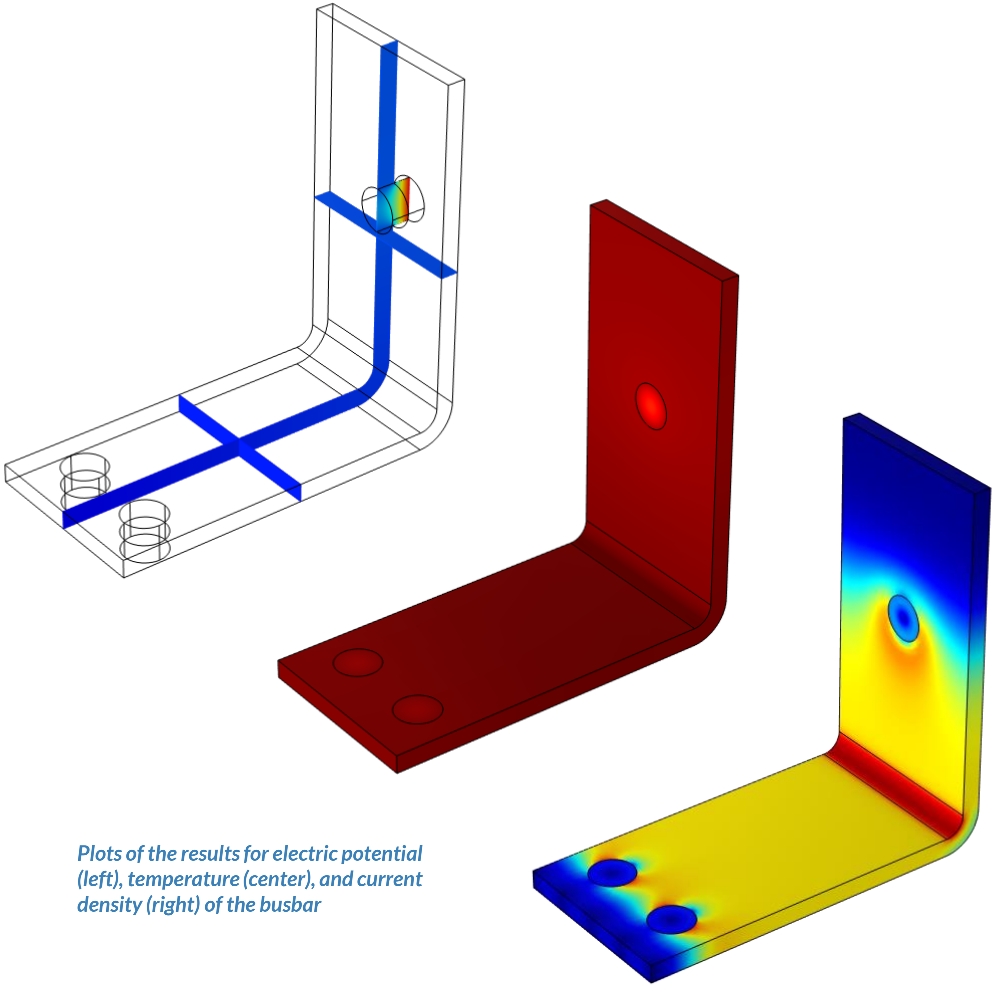
- Add a *Stationary* study
  - *Study 2*
- Change label for *Study 2* node to *Electrical-Thermal Analysis*
- Compute the model



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

Postprocess Results: Electrical–Thermal

- 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, *ec.normJ*
  - 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