Modeling Exercise

Define the physics for a model of a busbar
using the fully automatic approach

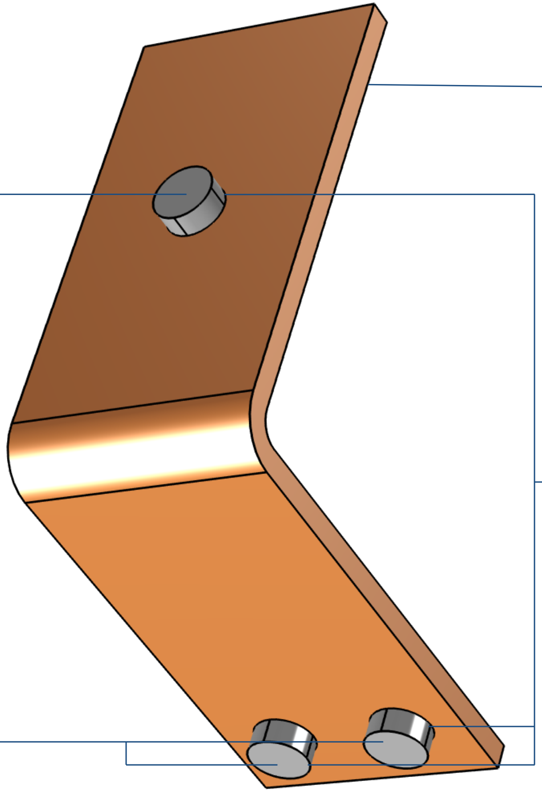
Introduction

- This model exercise demonstrates the concept of multiphysics modeling in COMSOL Multiphysics®
- Define the physics for the model using the fully automatic approach
  - Add the *Joule Heating* multiphysics interface
    - Completely streamlines defining the physics by automatically including the physics phenomena involved and the appropriate settings for the combination of physics phenomena involved
- 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



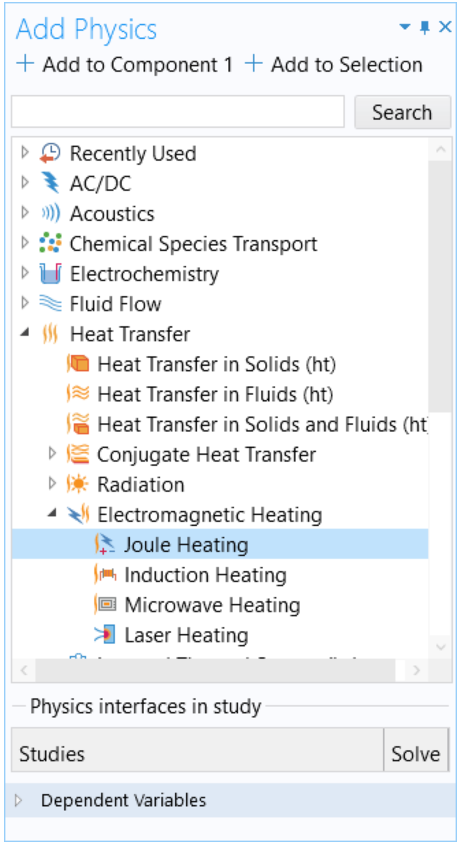
Details and specifications for the busbar model setup.

Fully Automatic Approach

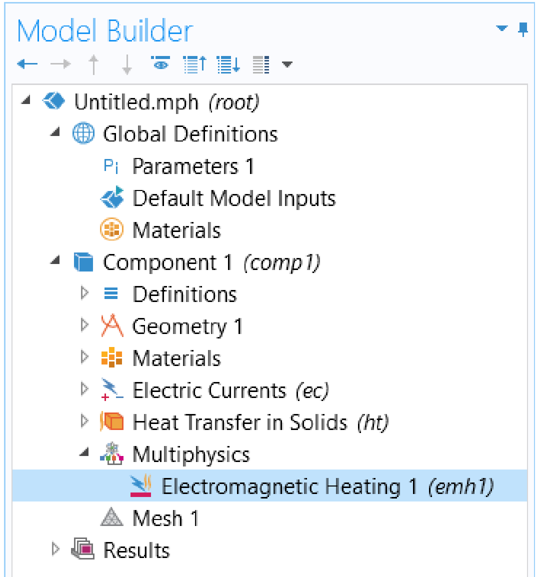
Define the physics for the model using a predefined multiphysics interface

Procedure:

1. Add the physics interface
2. Define the physics settings



The Add Physics window and the model tree after the multiphysics interface has been added.



COMSOL

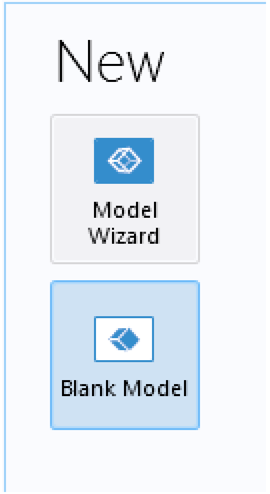
Modeling Workflow

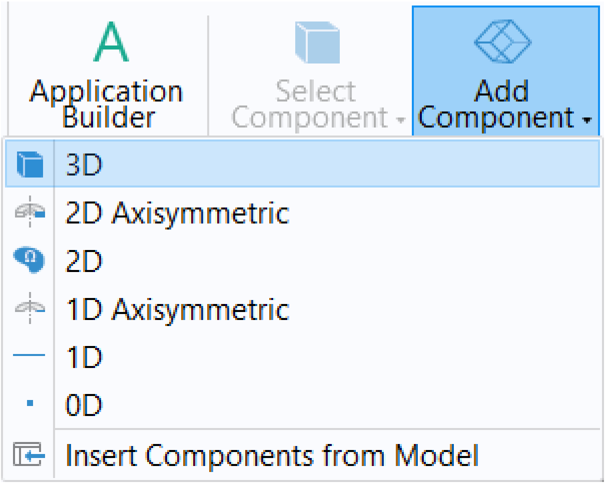
An outline of the steps used to set up, build, and compute this model to complete this modeling exercise is provided here.

1. Set up the model
2. Import geometry
3. Assign materials
4. Define the physics
    - Add *Joule Heating* multiphysics interface
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

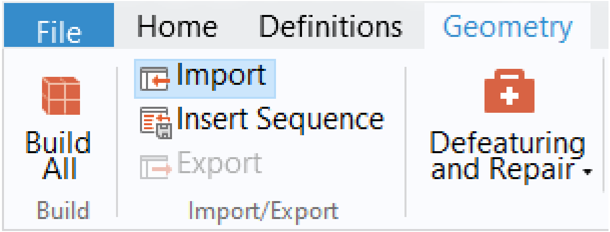


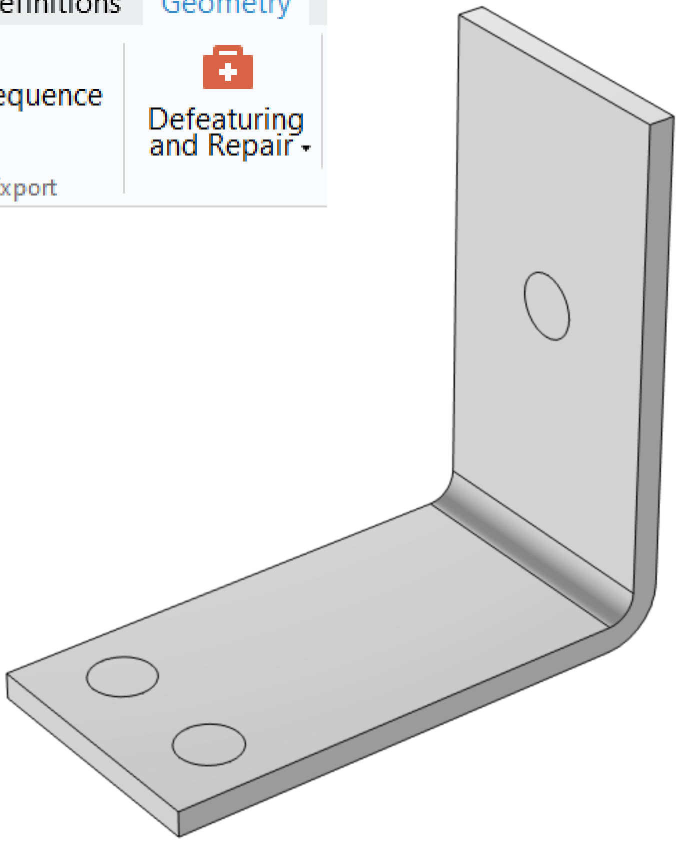


Screenshots of the steps performed to set up the model.

Import Geometry

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

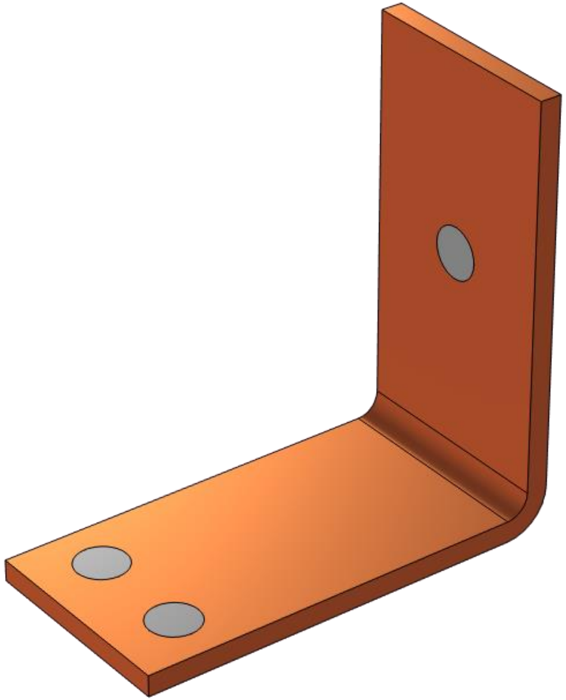




The Import button used and the busbar model geometry.

Assign Materials

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



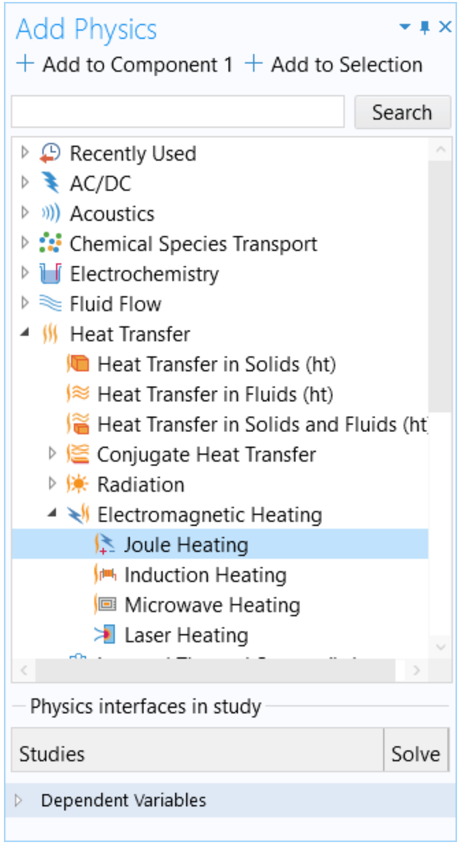
Busbar model with the Show Material Color and Texture *option enabled.*

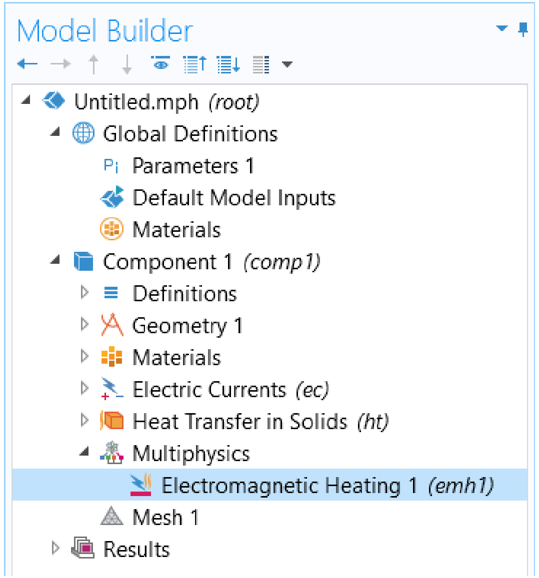
Fully Automatic Approach

Define the physics for the model using a predefined multiphysics interface

Procedure:

1. Add the physics interface
   - *Joule Heating* multiphysics interface
2. Define the physics settings
   - *Electric Currents* interface
   - *Heat Transfer in Solids* interface
   - *Electromagnetic Heating* multiphysics coupling



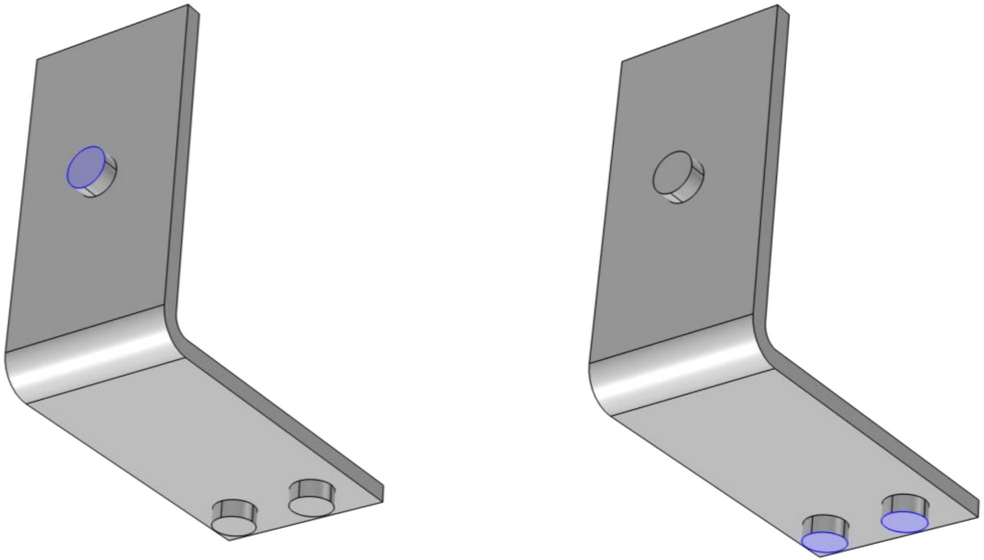


The Add Physics window and the model tree after the multiphysics interface has been added.

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



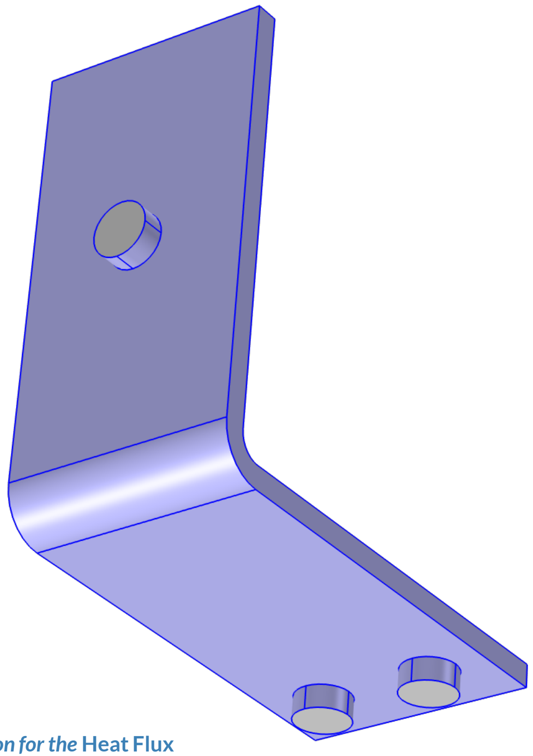
Geometry selection for the Electric Potential *(left) and* Ground *(right) boundary conditions.*

* = Refer to model specifications for values

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



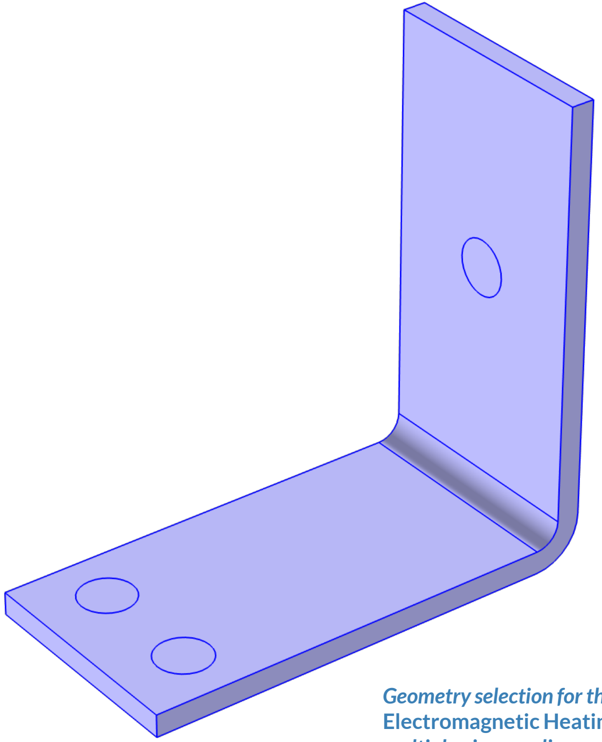
Geometry selection for the Heat Flux *boundary condition*

* = Refer to model specifications for values

MULTIPHYSICS SETTINGS

Electromagnetic Heating

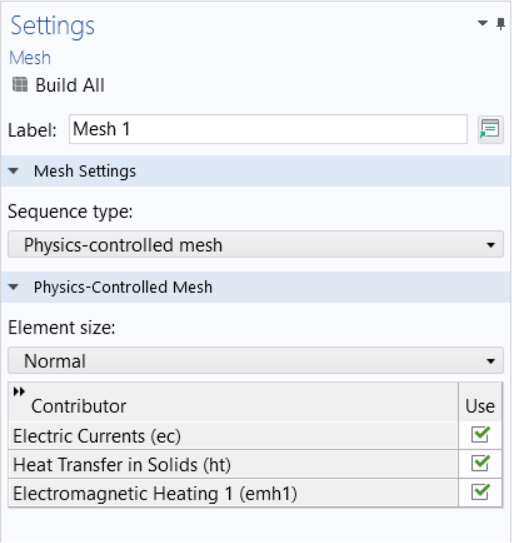
- Active in all domains
- Couples the *Electric Currents* and *Heat Transfer in Solids* physics interfaces
  - *Electric Currents*
    - Computes losses from passing electric current through the busbar
  - *Heat Transfer in Solids*
    - Incorporates resistive losses as a source of heat

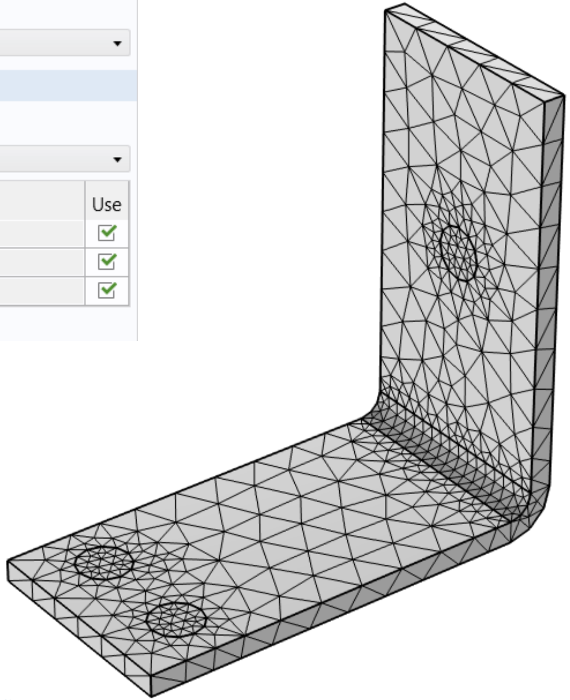


Geometry selection for the Electromagnetic Heating *multiphysics coupling node*

Build the Mesh

Build the mesh using the default settings

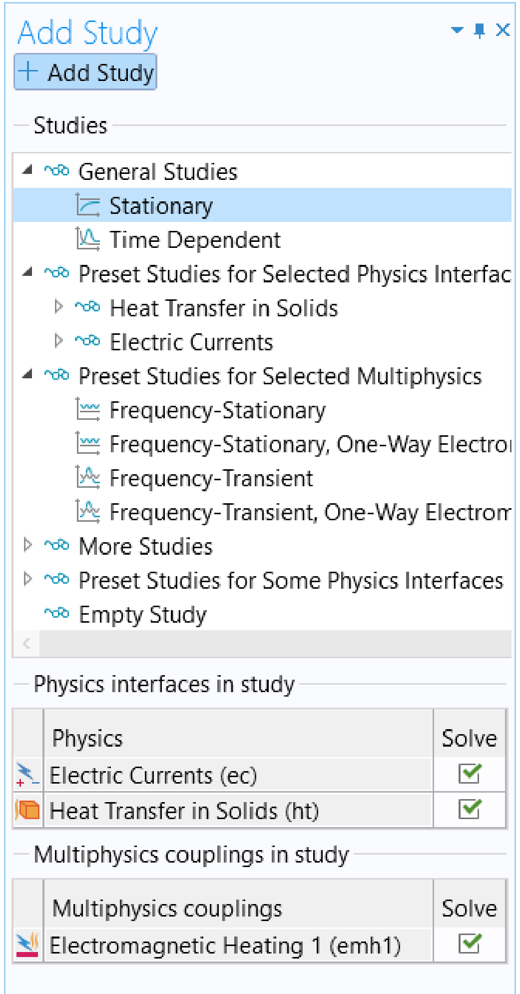




The settings used to generate the mesh for the busbar model, also pictured.

Run the Study

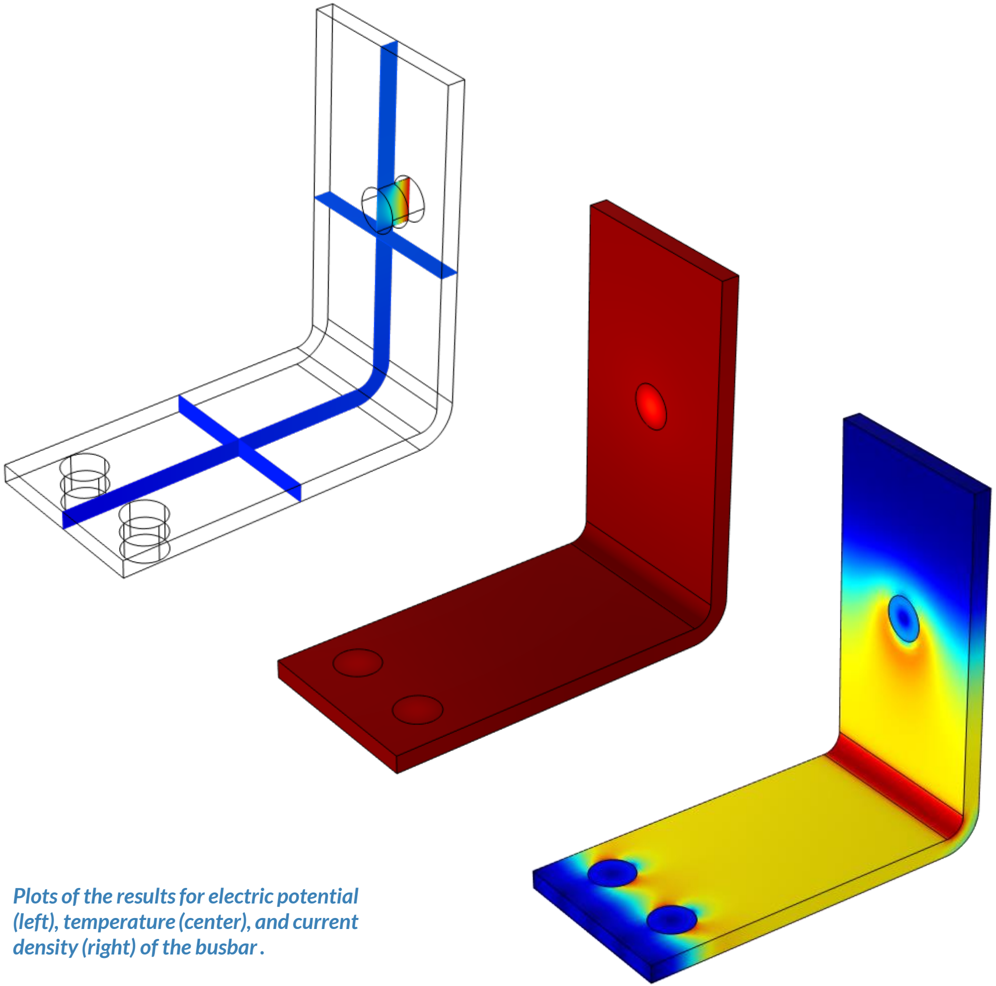
- Add a *Stationary* study
- Compute the model



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

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.