Modeling Exercise

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

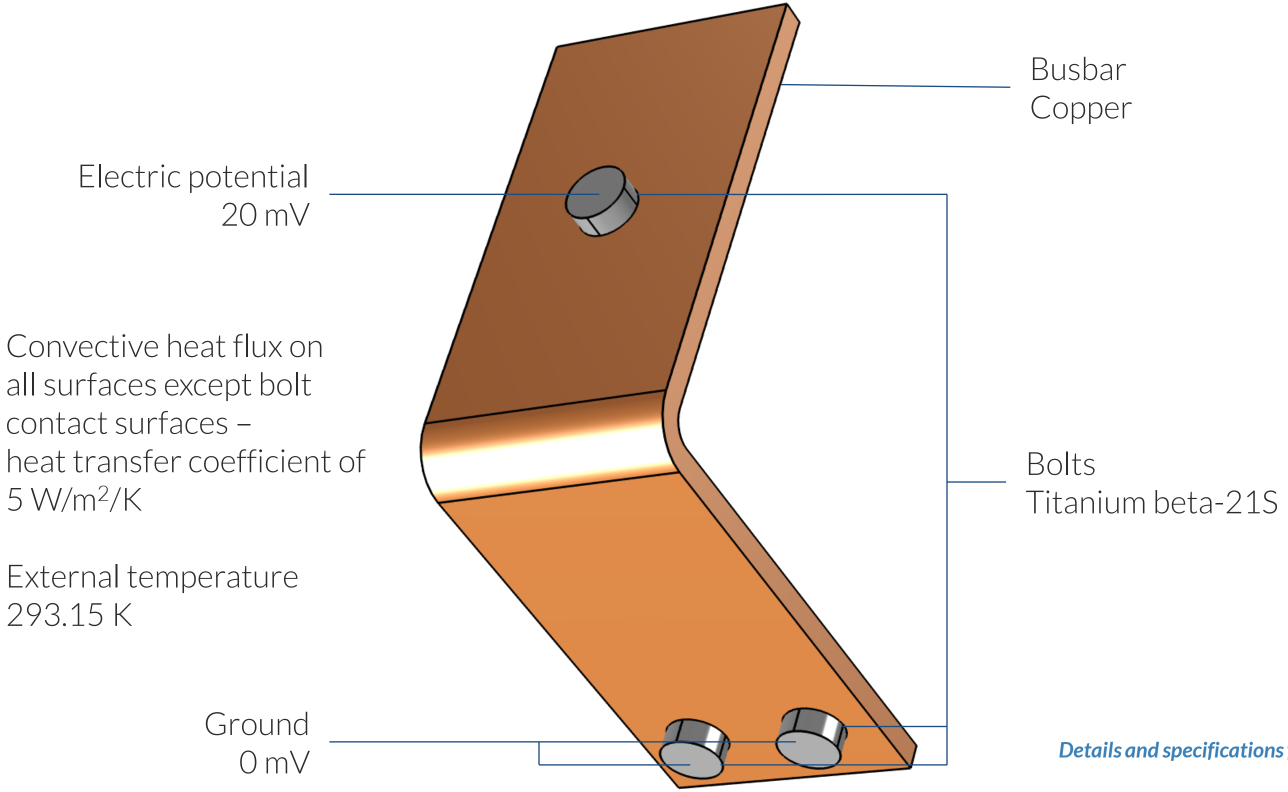
Introduction

- 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



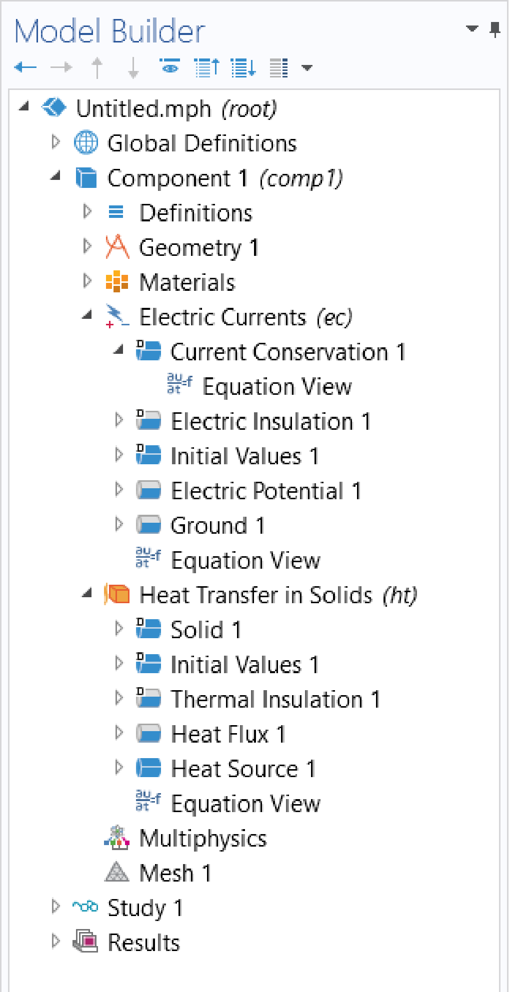
Details and specifications for the busbar model setup

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

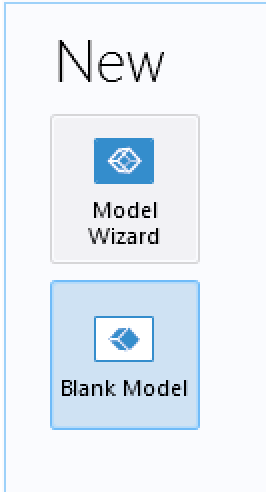
Modeling Workflow

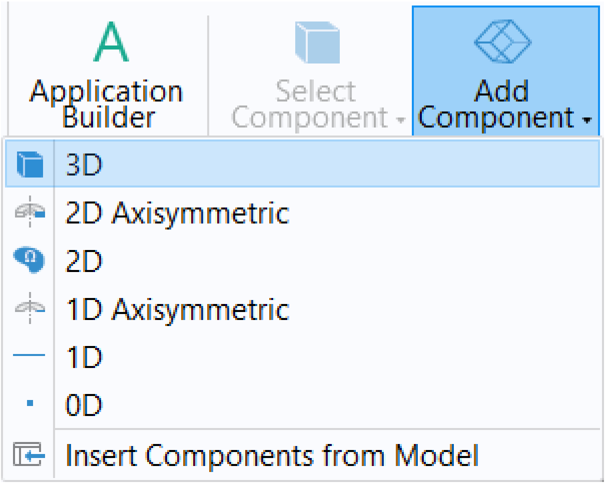
An outline of the steps used to set up, build, and compute this model to complete this modeling exercise is provided here:

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

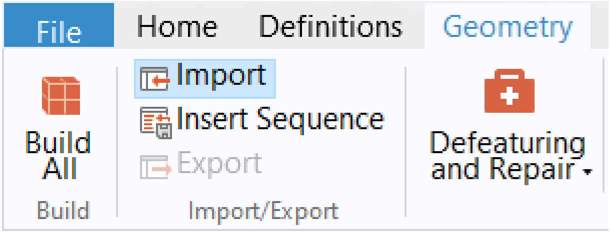


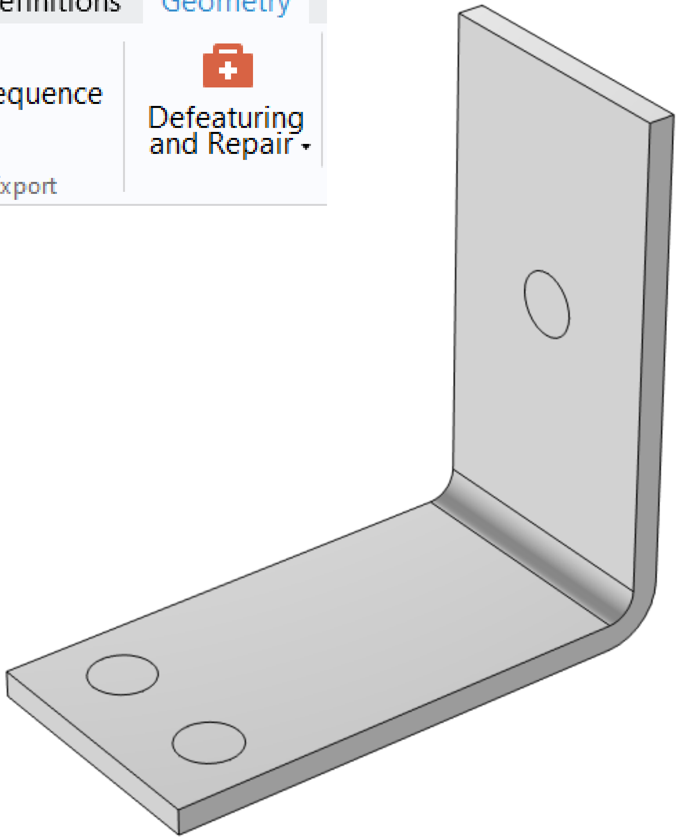


Screenshots of the steps performed to set up the model

Import Geometry

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

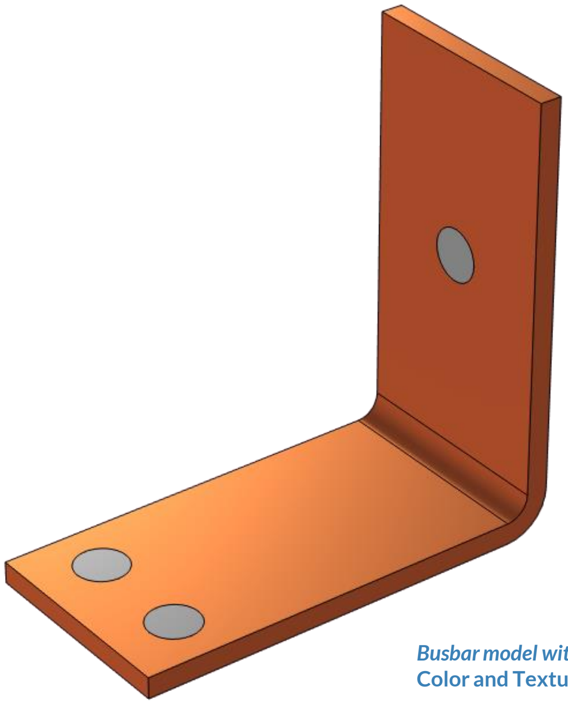




The Import button used and the busbar model geometry

Assign Materials

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



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

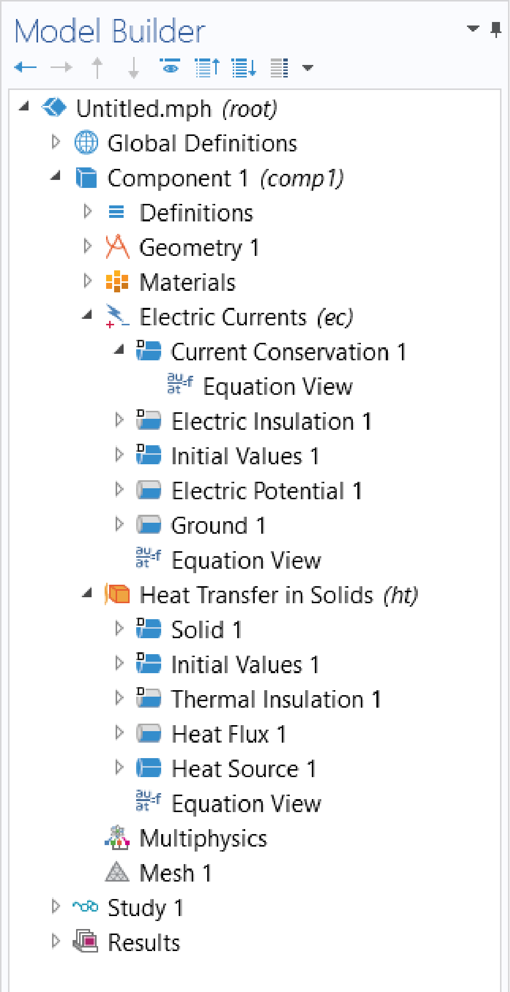
Manual Approach with User-Defined Couplings

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

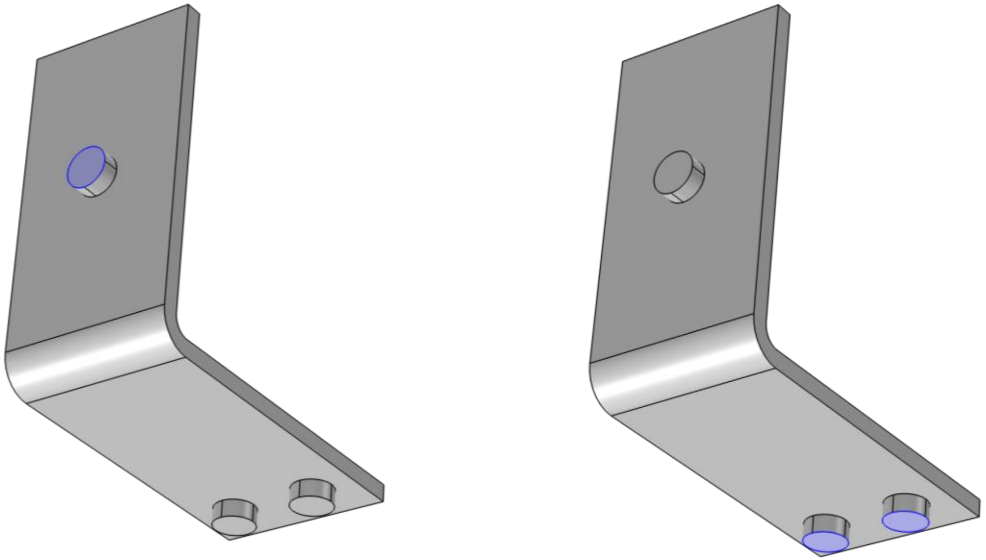


PHYSICS SETTINGS

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



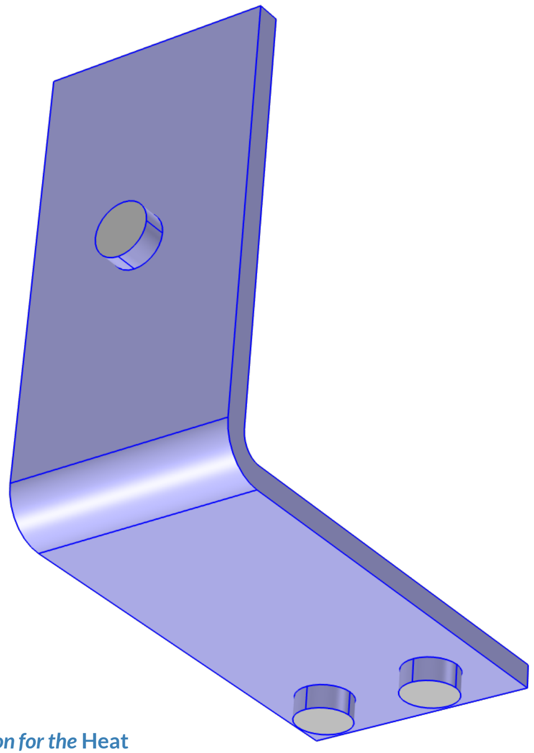
Geometry selection for the Electric Potential *(left) and* Ground *(right) boundary conditions*

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

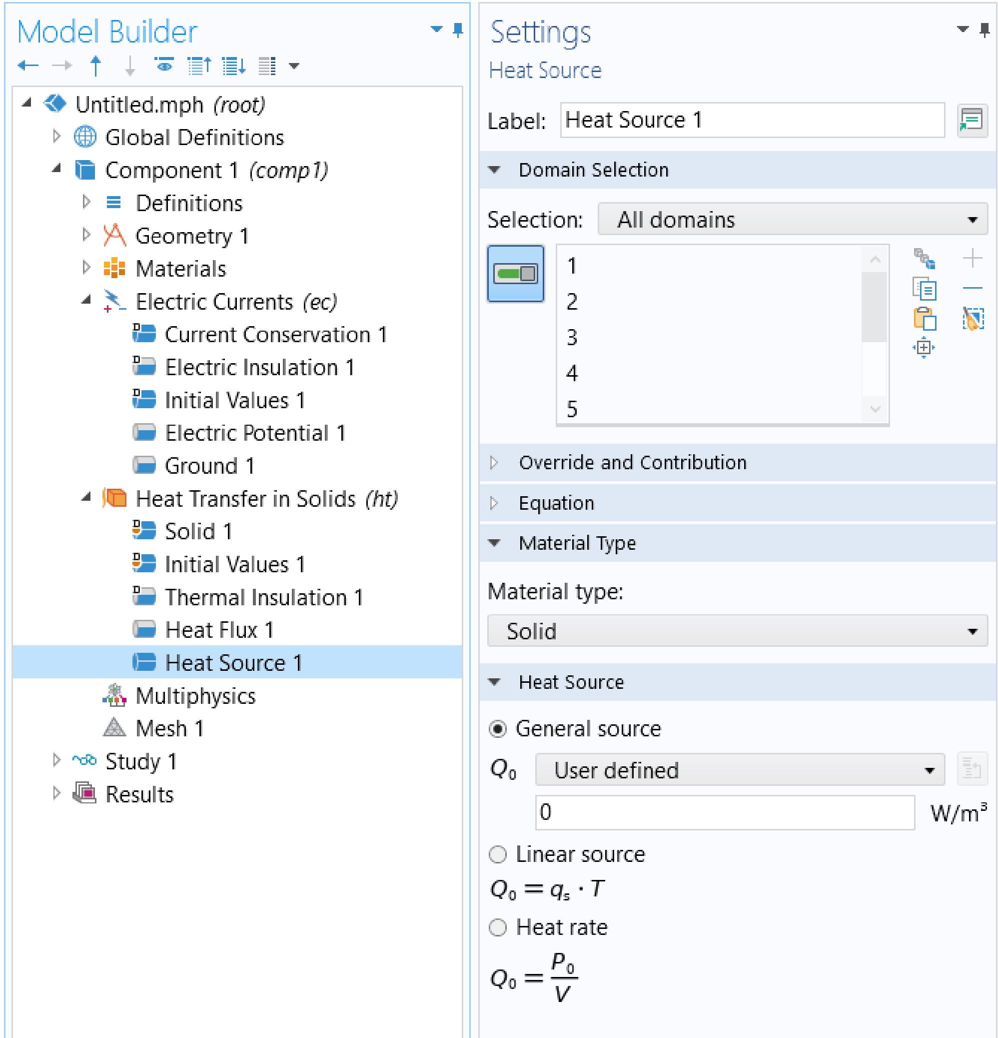


Geometry selection for the Heat Flux *boundary condition*

MULTIPHYSICS SETTINGS

Electromagnetic Heating

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

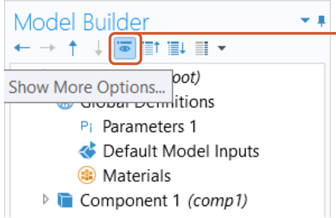


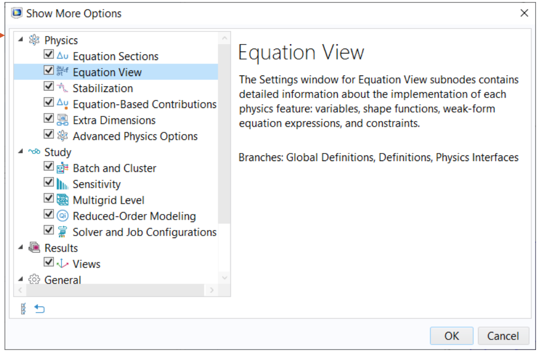
The Settings window for the Heat Source node

MULTIPHYSICS SETTINGS

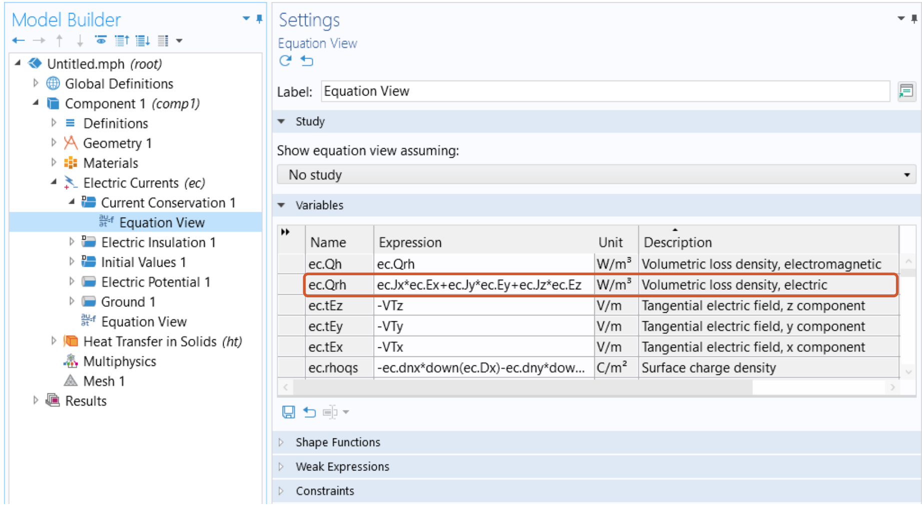
Electromagnetic Heating

- 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





The Equation View node for the Current Conservation node



MULTIPHYSICS SETTINGS

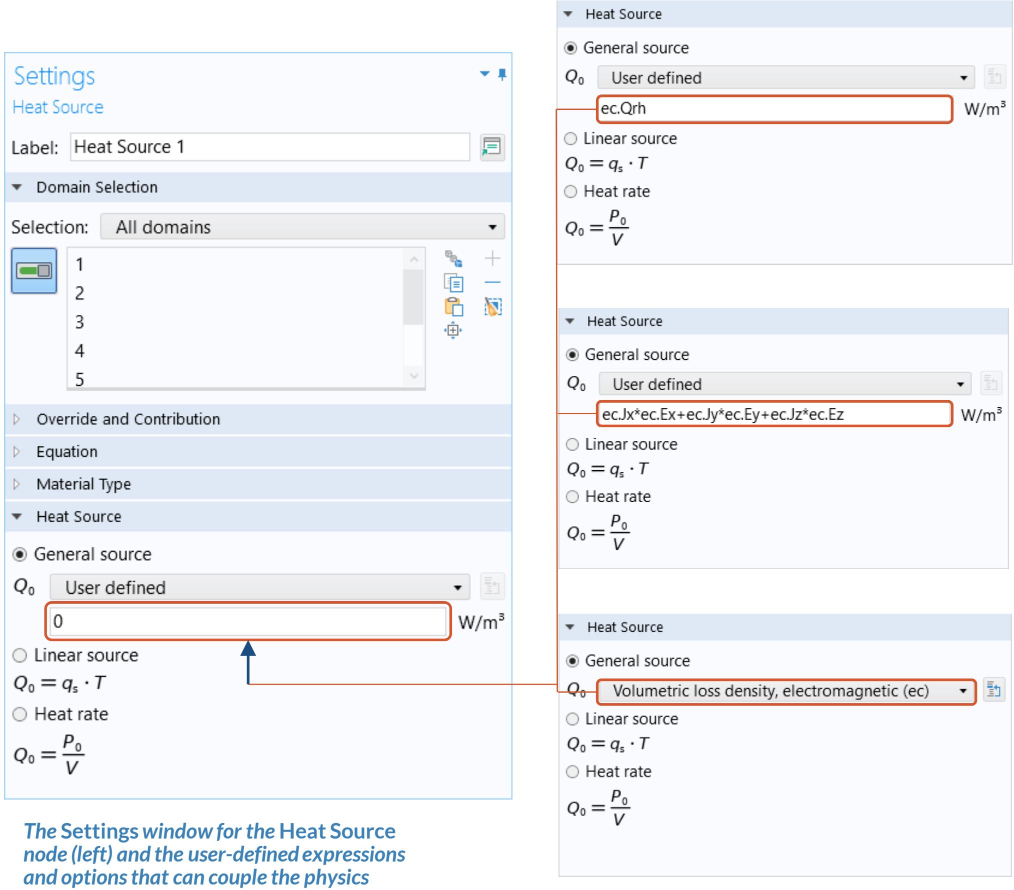
Electromagnetic Heating

- 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:

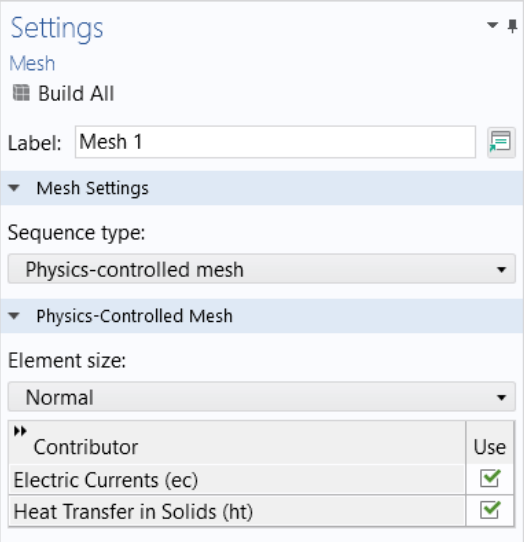
    Volumetric loss density, electromagnetic (ec)



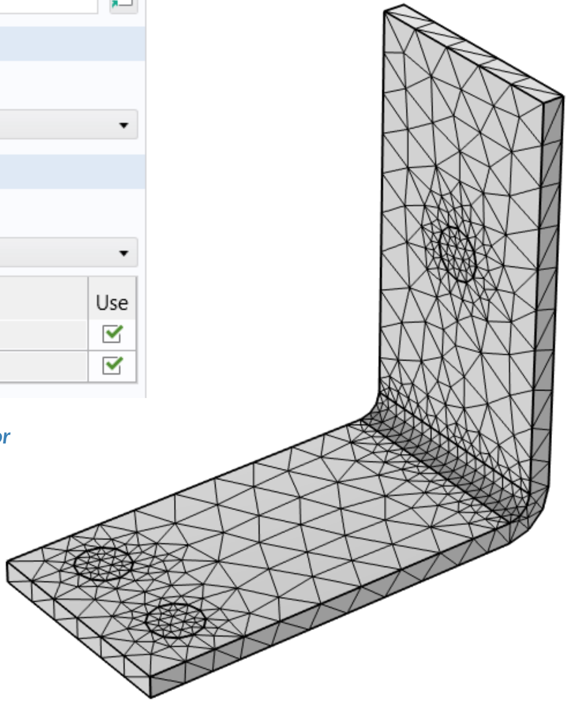
The Settings window for the Heat Source node (left) and the user-defined expressions and options that can couple the physics (right)

Build the Mesh

Build the mesh using the default settings

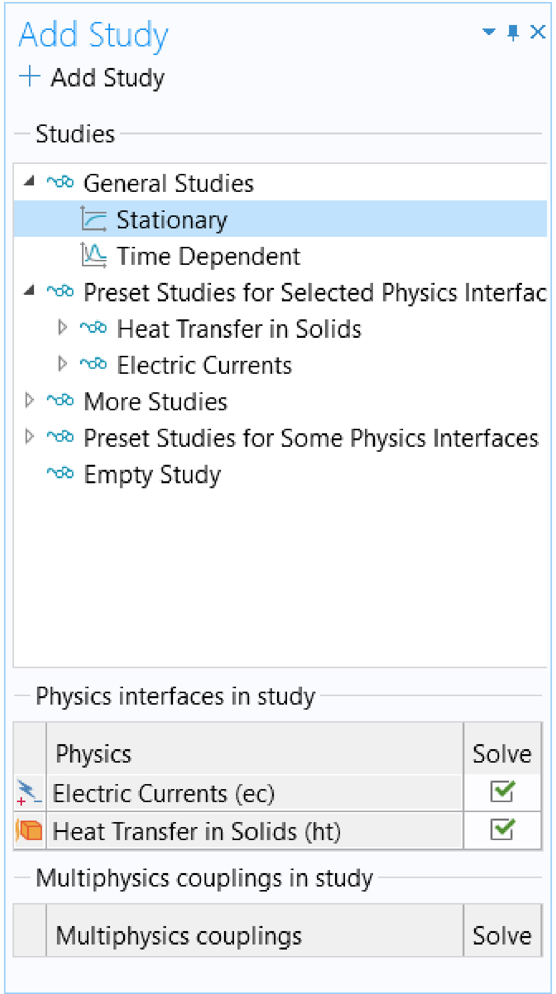


The settings used to generate the mesh for the busbar model, also pictured



Run the Study

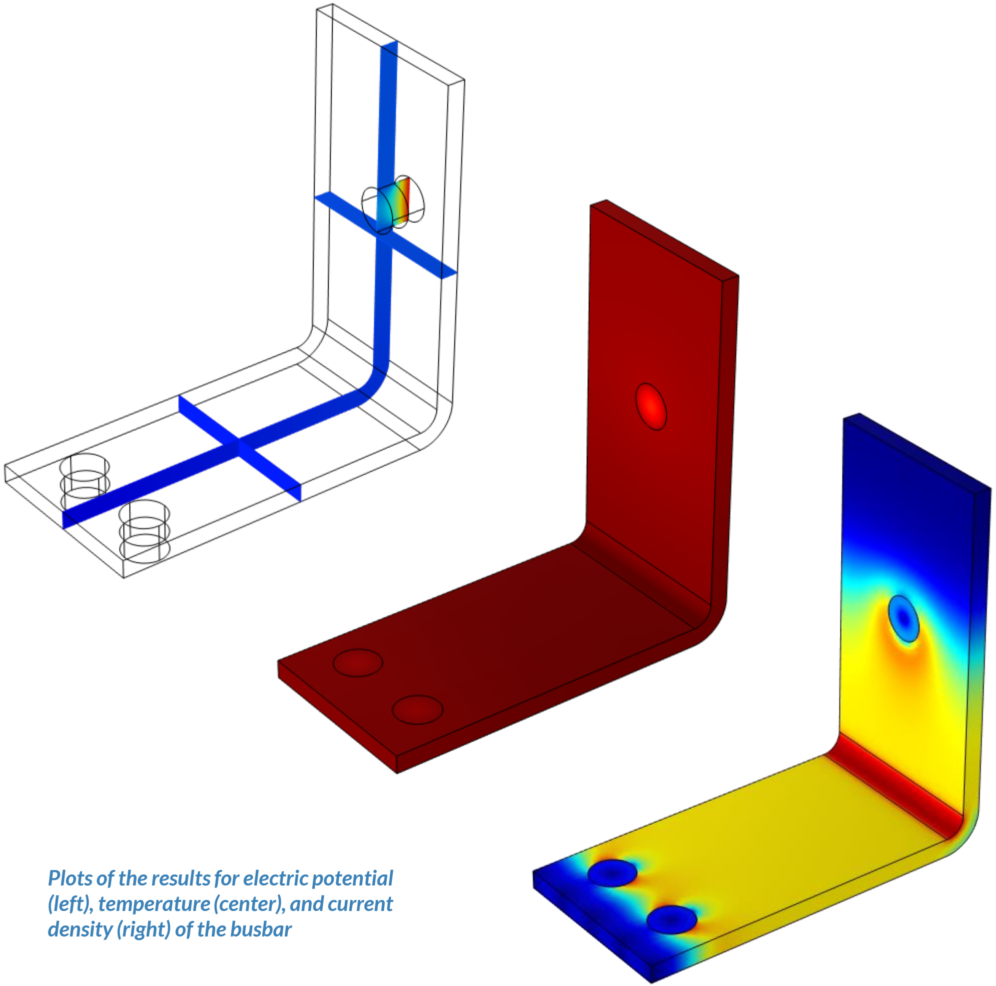
- Add a *Stationary* study
- Compute the model



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