Modeling Exercise

Define the physics for a model of a busbar using the manual approach with predefined couplings

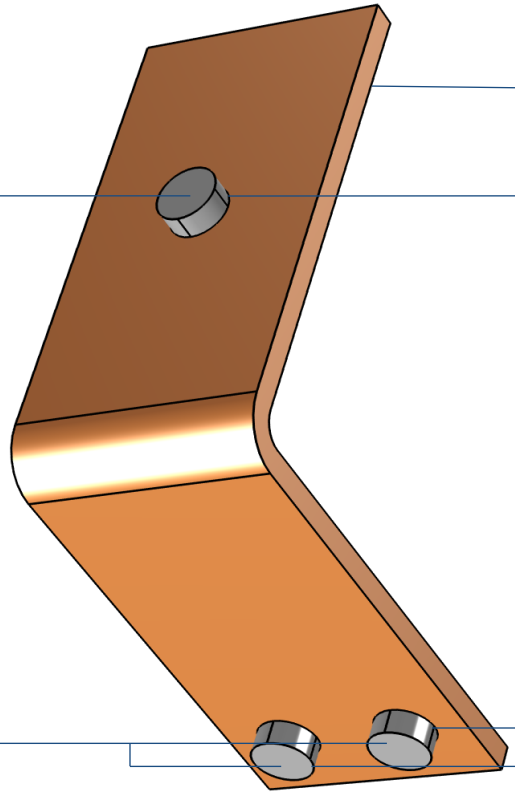
Introduction

- 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



Busbar
Copper

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

Bolts
Titanium beta-21S

Ground
0 mV

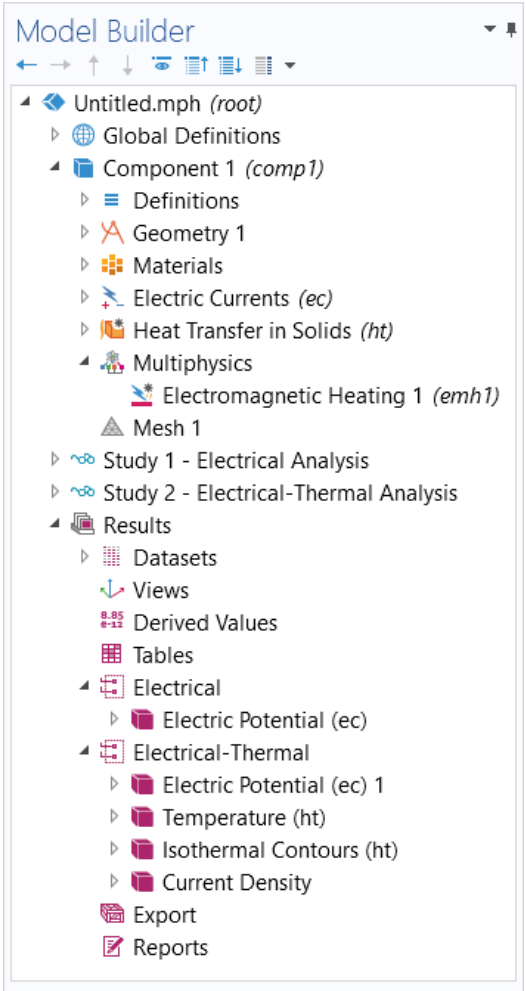
Details and specifications for the busbar model setup.

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