Modeling Exercise

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

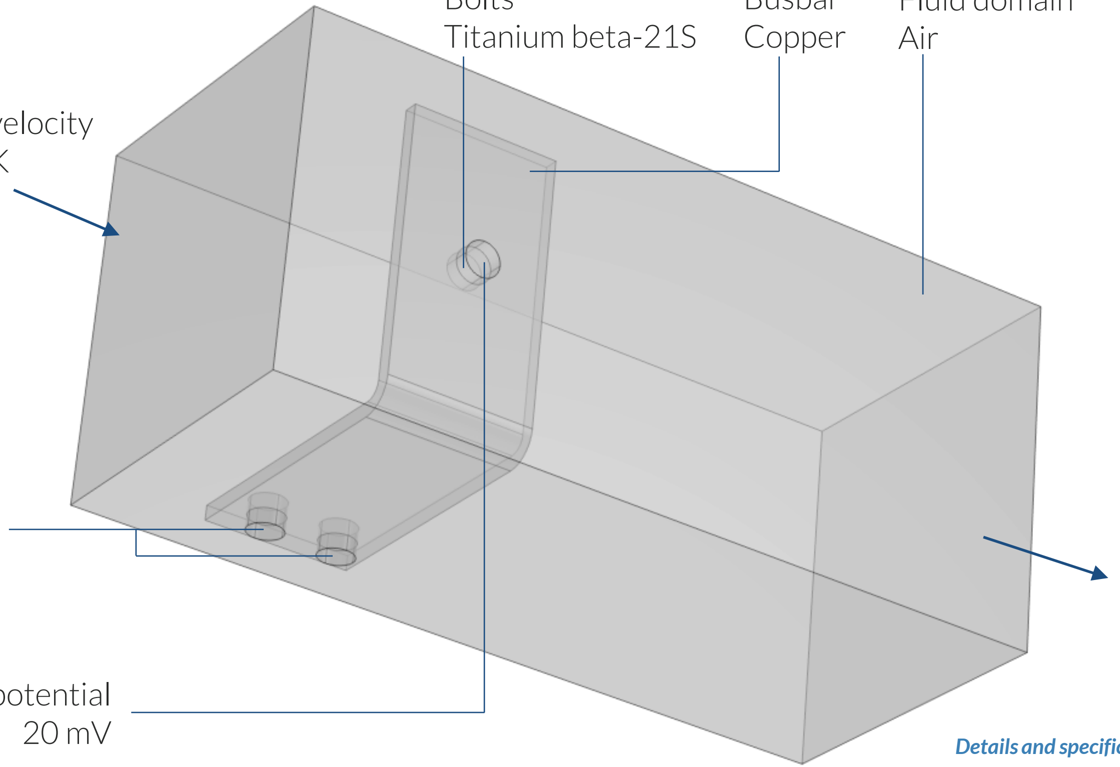
Introduction

- This modeling 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* coupling for the Joule heating, followed by another multiphysics simulation including the *Laminar Flow (spf)* interface and *Nonisothermal Flow* coupling for the nonisothermal flow
  - Note: This approach can be implemented in different ways, both with respect to the physics interfaces involved and the number of studies used
    - This exercise demonstrates one of these such ways
- Important information for setting up the model can be found in the model specifications
  - 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, which in turn causes the temperature of the busbar to rise
  - An instance of the Joule heating effect
- The busbar is cooled by air flowing over its surfaces, through convection
  - An instance of nonisothermal flow
- Results include the electric potential, temperature distribution, velocity field, and pressure distribution
  - Plot of the current density of the busbar assembly is manually generated

Model Specifications



Symmetry at boundaries parallel to fluid flow, since flow outside these boundaries is parallel to the flow inside boundaries

Details and specifications for the busbar convective cooling model.

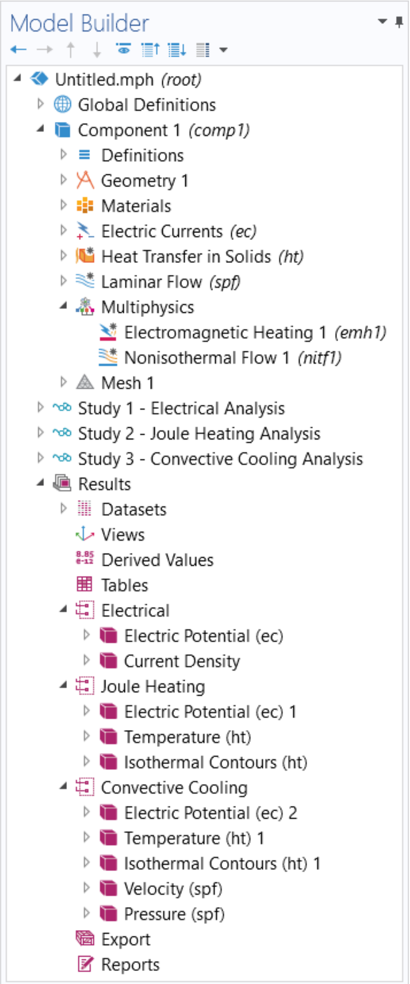
Manual with Predefined Couplings Approach

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 box 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

Joule Heating Analysis

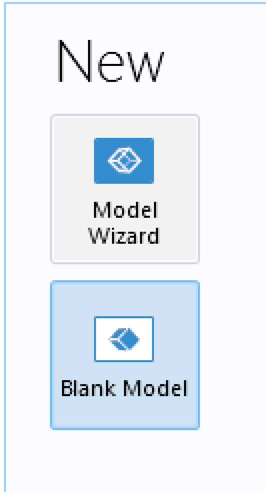
1. Define the physics
   - Add *Heat Transfer in Solids* interface
   - Add *Electromagnetic Heating* multiphysics coupling
2. Run the study
   - Add *Stationary* study
3. Check the results

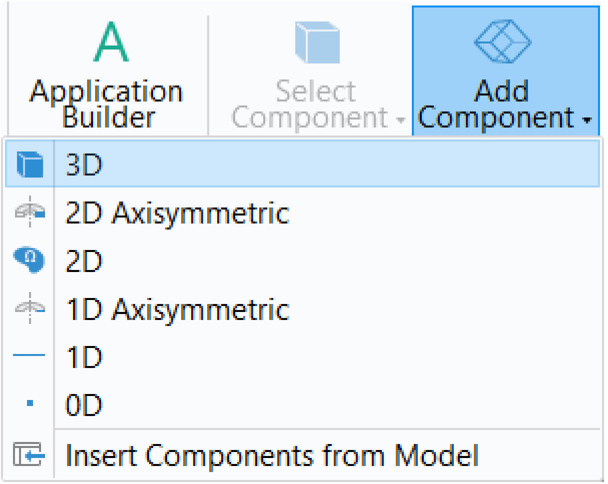
Convective Cooling Analysis

1. Define the physics
   - Add *Laminar Flow (spf)* interface
   - Add *Nonisothermal Flow* multiphysics coupling
2. Run the study
   - Add *Stationary* study
3. Check the results

Model Setup

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

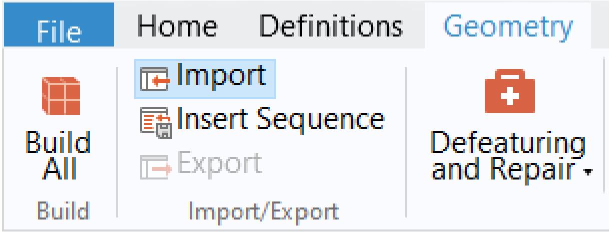


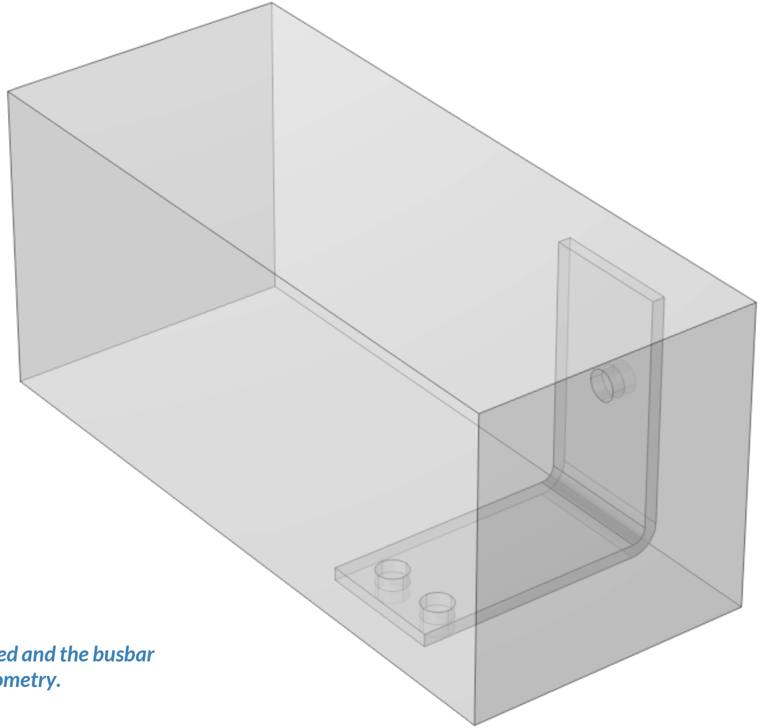


Screenshots of the steps performed to set up the model.

Import Geometry

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

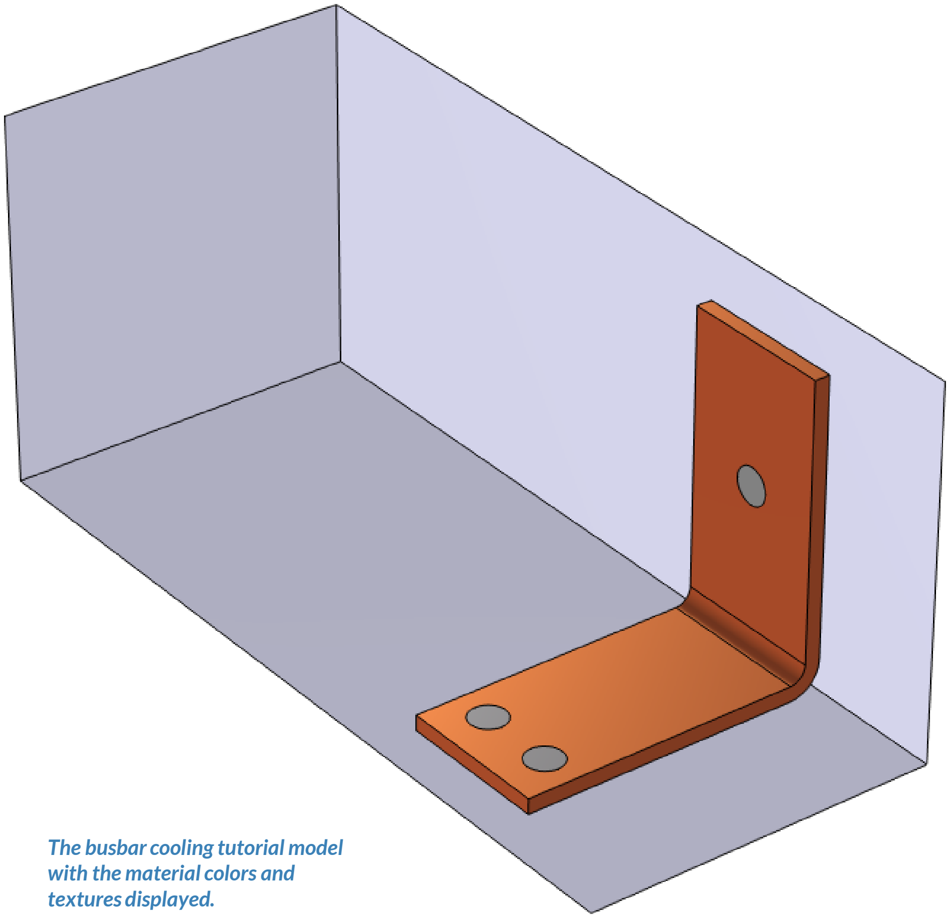




The Import button used and the busbar box tutorial model geometry.

Assign Materials

- Busbar
  - Apply *Copper*
- Bolts
  - Apply *Titanium beta-21S*
- Fluid domain
  - Apply *Air*



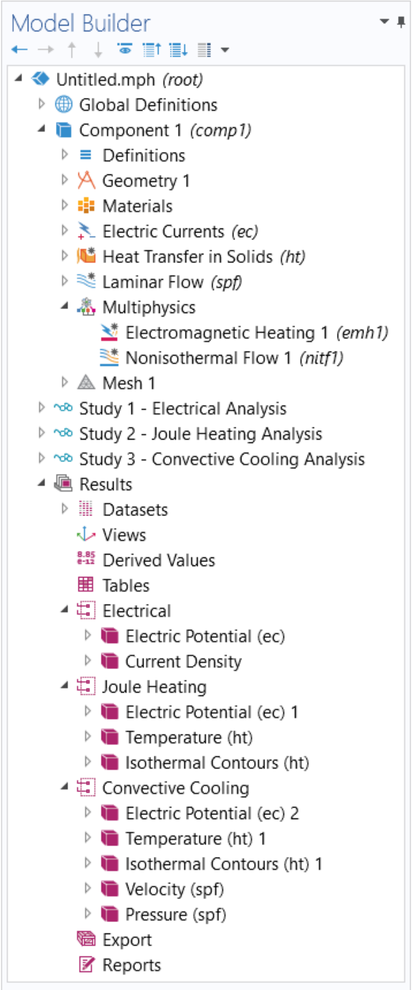
The busbar cooling tutorial model with the material colors and textures displayed.

Manual with Predefined Couplings Approach

Perform three stationary studies with the first for the electric currents, the second including heat transfer, and the third including fluid flow

Procedure:

1. Electrical analysis
   - Add and define settings for the *Electric Currents* interface
2. Joule heating analysis
   - Add and define settings for the *Heat Transfer in Solids* interface
   - Add the *Electromagnetic Heating* multiphysics coupling
3. Convective cooling analysis
   - Add and define the settings for the *Laminar Flow (spf)* interface
   - Add the *Nonisothermal Flow* multiphysics coupling



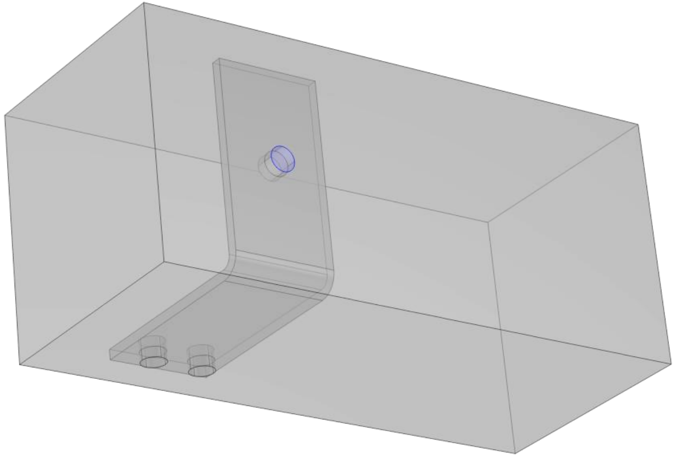
The model tree for the busbar tutorial model when the manual approach with predefined couplings has been used.

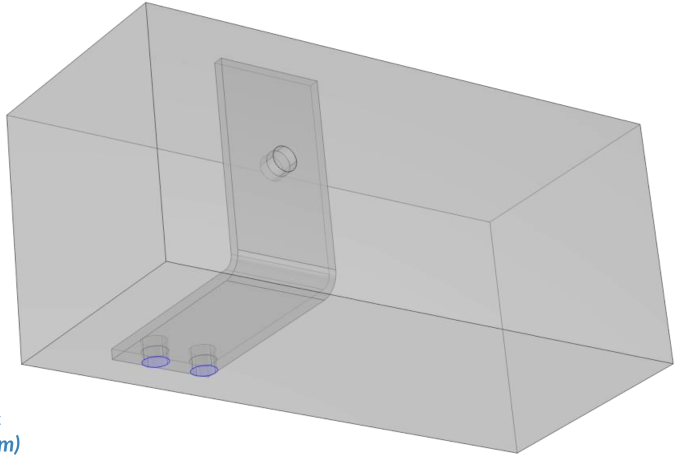
PHYSICS SETTINGS

Electric Currents

- Check that the *Domain Selection* for the interface includes the busbar and bolt domains only
- 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

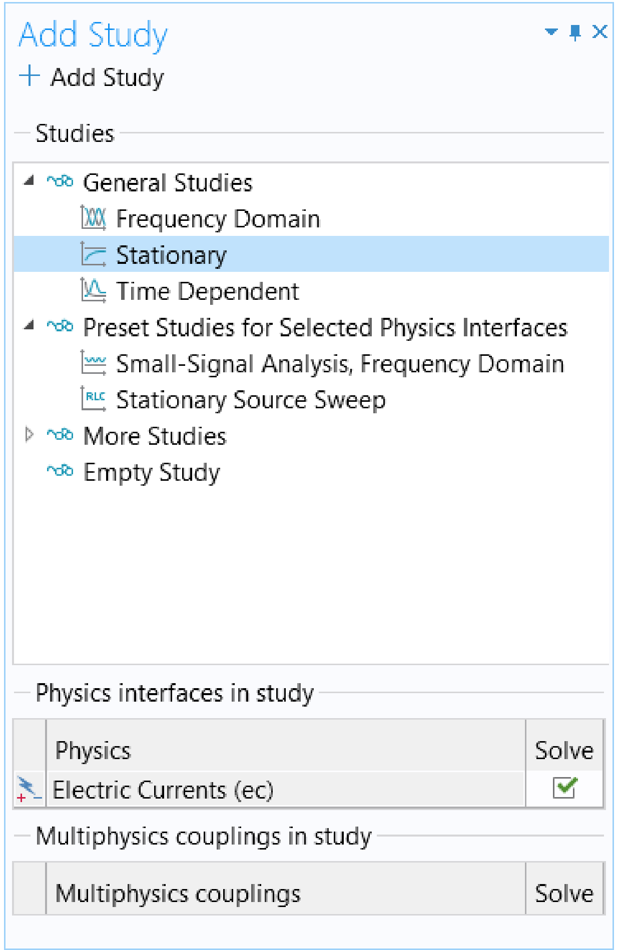




Geometry selection for the Electric Potential *(top) and* Ground *(bottom) boundary conditions.*

Run the Study: Electrical

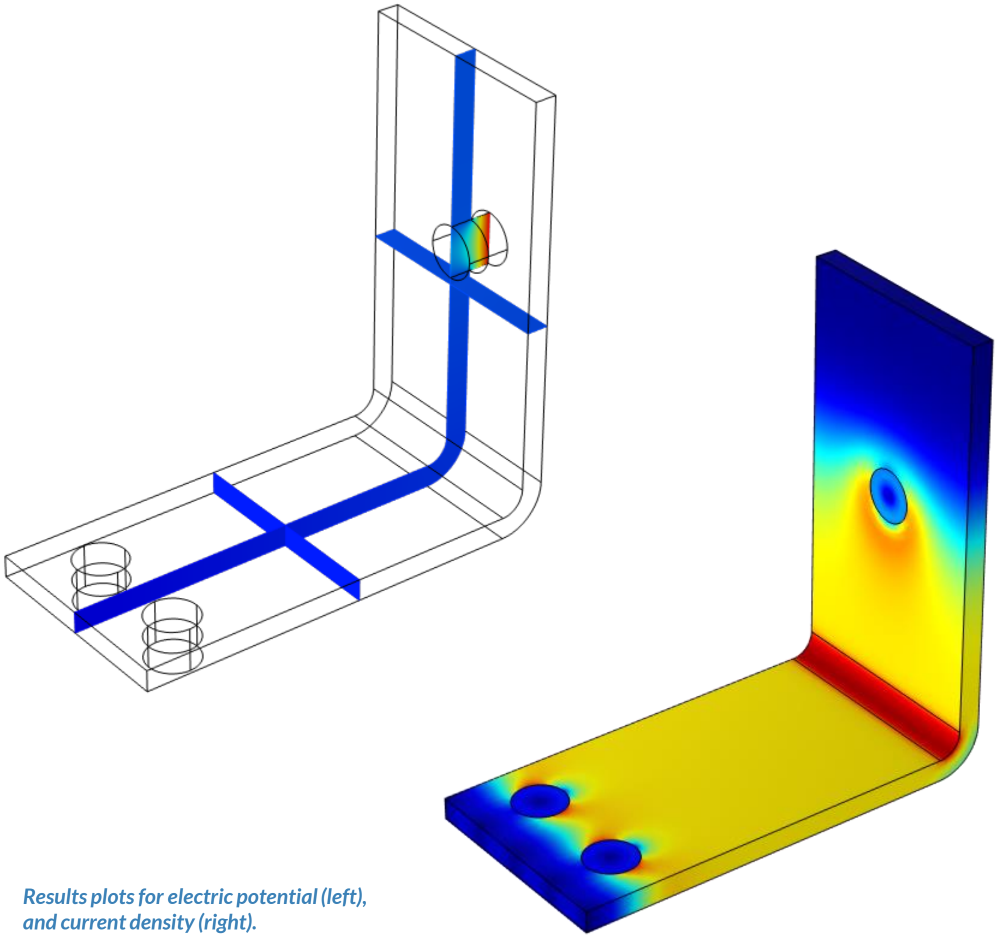
- Add a *Stationary* study
  - *Study 1*
- Change label for *Study 1* node to *Electrical Analysis*
- Compute the model



Settings for the first Stationary study added to the model.

Postprocess Results: Electrical

- Default plots generated by the software
  - Electric Potential
- Create a plot of the current density
  - Add a new *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*



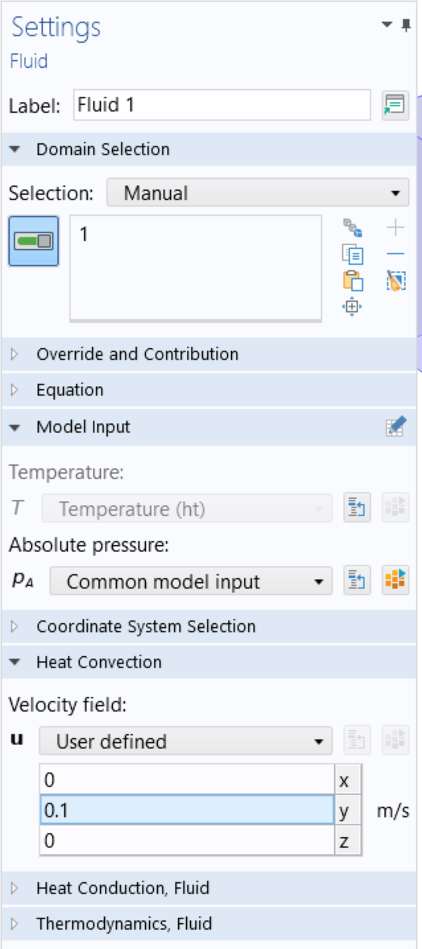
Results plots for electric potential (left), and current density (right).

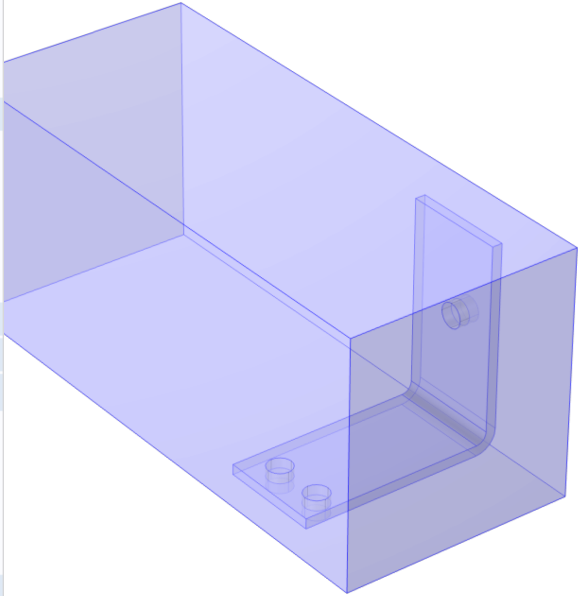
PHYSICS SETTINGS

Heat Transfer in Solids

- Active in all domains
- Add *Fluid* domain feature*
  - Select fluid domain geometry
  - Under *Heat Convection* section, specify a *User-defined* velocity field
- Add *Temperature* boundary condition*
  - Defines temperature at inlet
- Add *Outflow* boundary condition
  - Defines outlet for heat transfer

* = Refer to model specifications for values



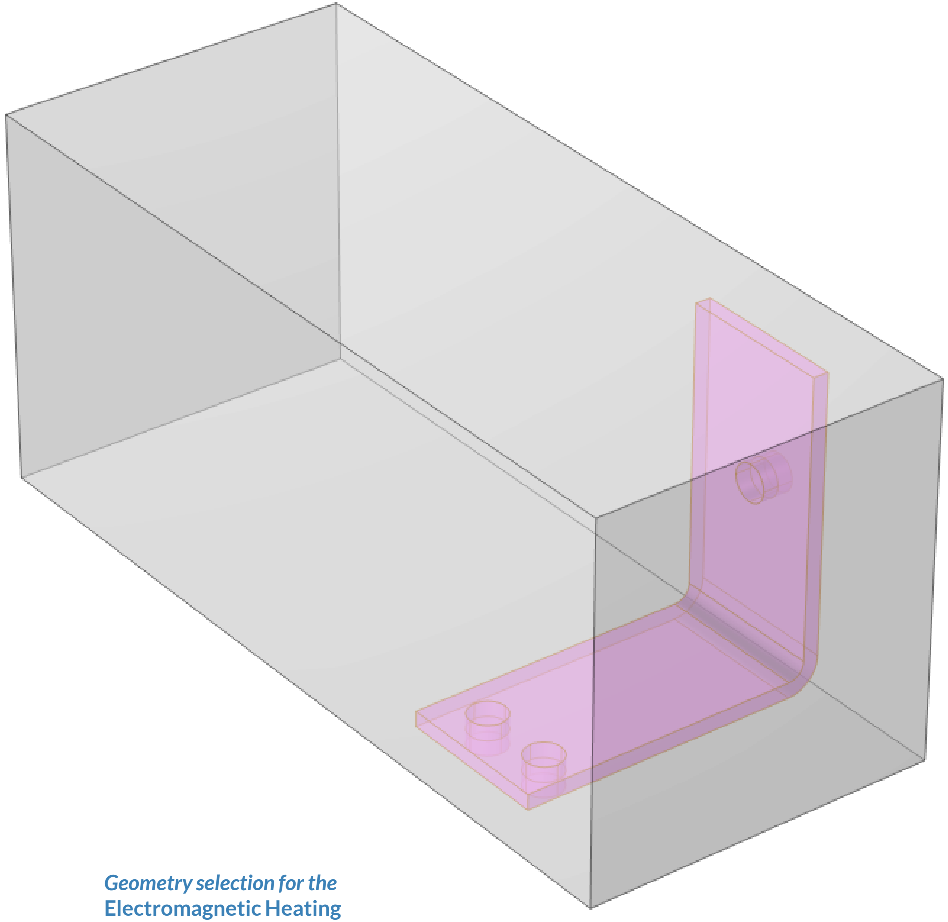


The Settings window and geometry selection for the Fluid *domain feature.*

MULTIPHYSICS SETTINGS

Electromagnetic Heating

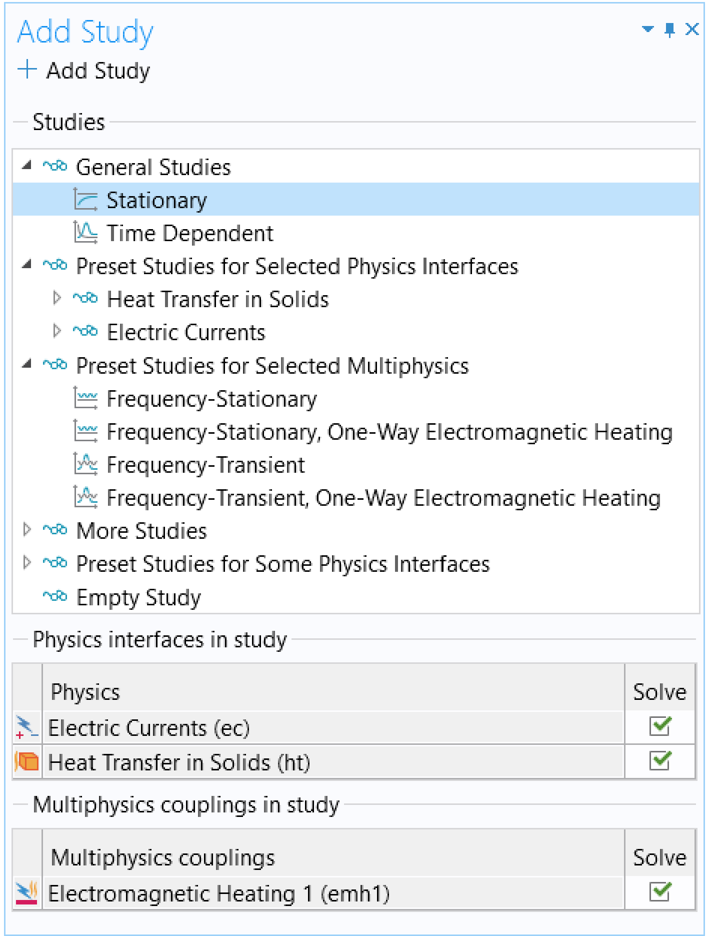
- Active in busbar and bolt domains only by default
- Couples the *Electric Currents* and *Heat Transfer in Solids* physics interfaces
  - *Electric Currents*
    - Computes losses from passing electric current through the busbar
  - *Heat Transfer in Solids*
    - Incorporates resistive losses as a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



Geometry selection for the **Electromagnetic Heating** ***multiphysics coupling node***

Run the Study: Electrical-Thermal

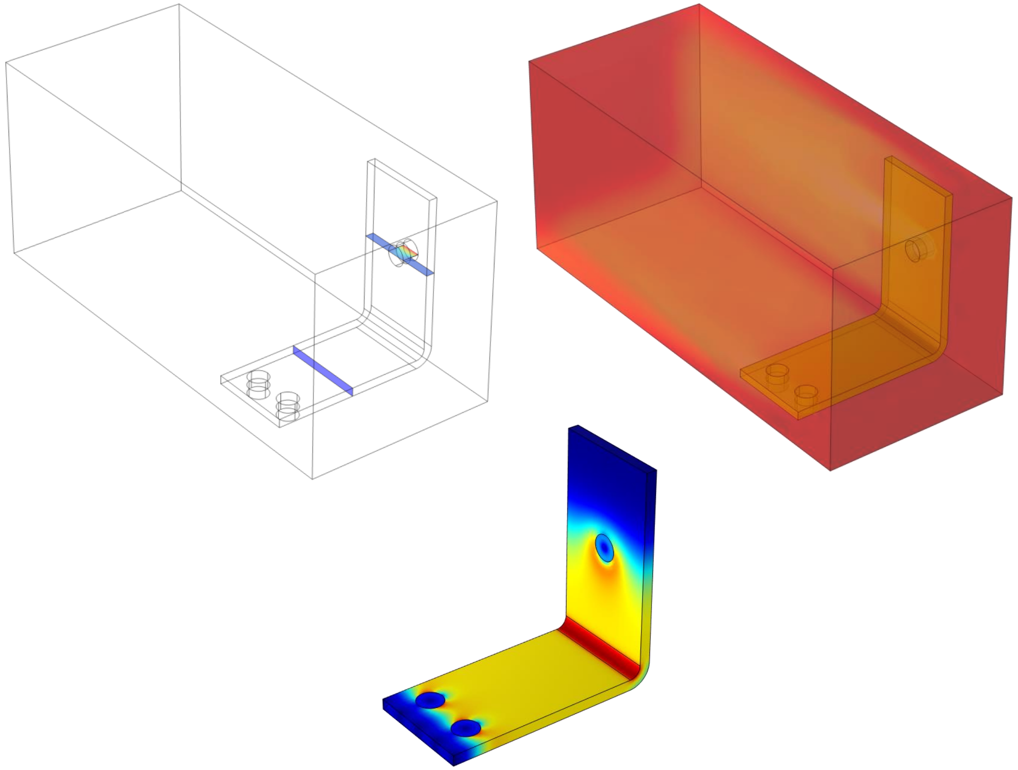
- Add a *Stationary* study
  - *Study 2*
- Change label for *Study 2* node to *Joule Heating Analysis*
- Compute the model



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 a plot of the current density
  - Duplicate the *Current Density* plot and rename it *Current Density 2*
  - Change the *Dataset* to *Study 2/Solution 2 (sol 2)*

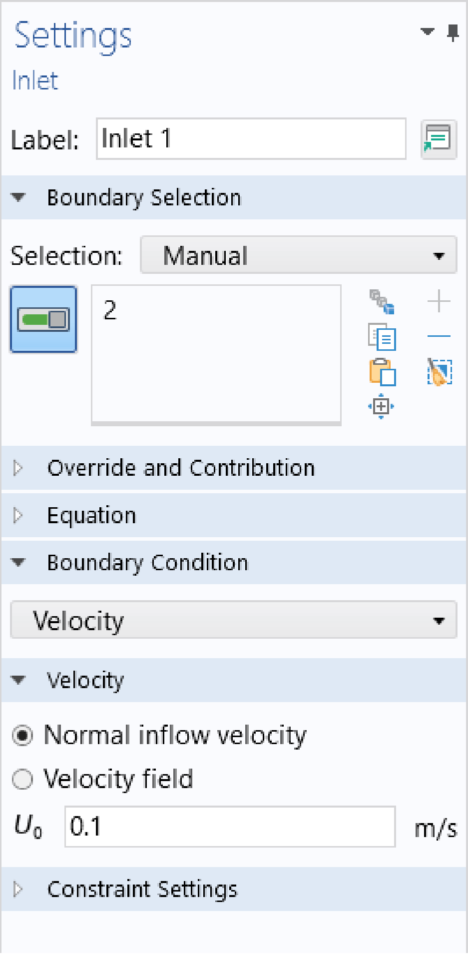


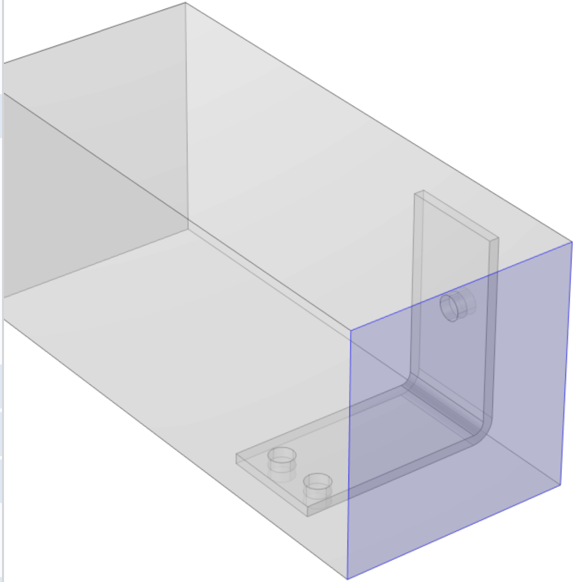
Plots of the results for electric potential (left), temperature (top right), and current density (bottom right).

Laminar Flow

- Check that the *Domain Selection* for the interface includes the fluid domain only
- Add *Inlet* boundary condition*
  - Defines flow velocity into domain
- Add *Outlet* boundary condition
  - Defines pressure at outlet
- Add *Symmetry* boundary condition
  - Defines symmetry boundaries

* = Refer to model specifications for values



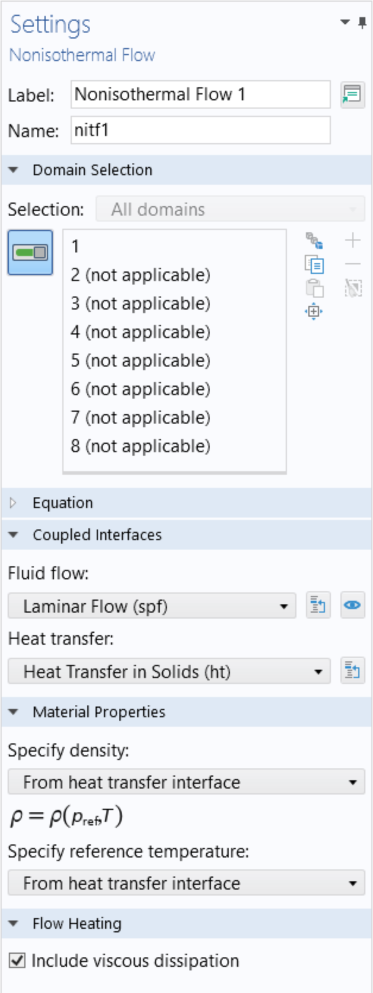


The Settings window and geometry selection for the Inlet boundary condition.

MULTIPHYSICS SETTINGS

Nonisothermal Flow

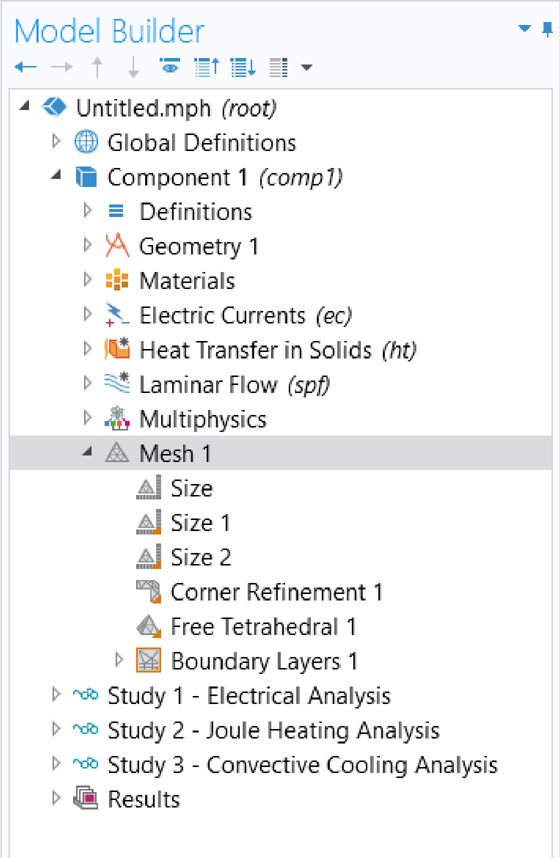
- Active in fluid domain only by default
- Couples the *Heat Transfer in Solids* and *Laminar Flow (spf)* physics interfaces
  - *Heat Transfer in Solids*
    - Incorporates the pressure distribution and velocity field from the fluid flow interface
  - *Laminar Flow*
    - Incorporates the temperature field and defines the density of the fluid, among other fluid properties

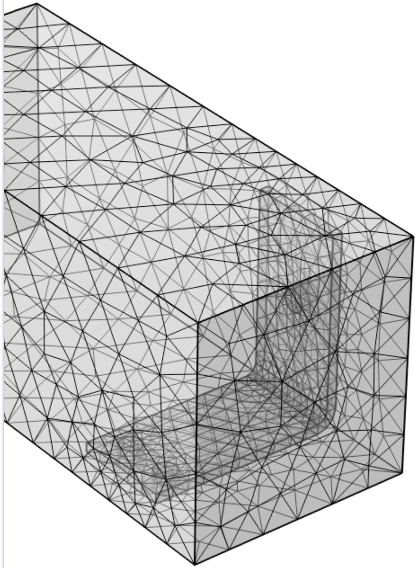


Settings ***window for the*** **Nonisothermal Flow** ***multiphysics coupling.***

Build the Mesh

- Build and then edit the mesh to reduce the computational resources required when re-computing the model
- *Mesh 1* node
  - Build the mesh with the default settings
  - Change *sequence type* to *User-controlled mesh*
  - Change *Size 1* node's *Element Size* to *Extremely Coarse*
    - Fluid domain
  - Change *Size 2* node's *Element Size* to *Normal*
    - Busbar and bolt boundaries in fluid domain

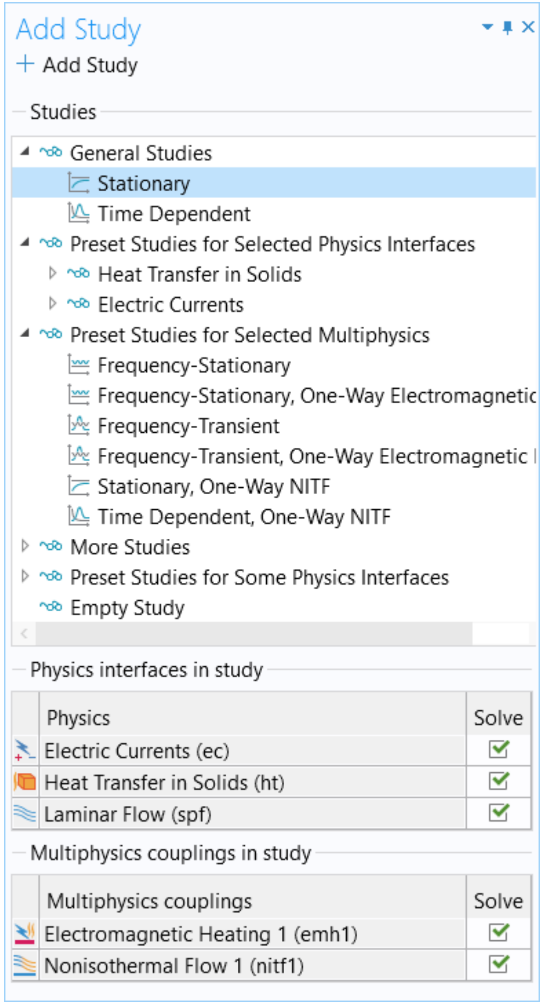




The meshing sequence and resultant mesh generated for the convective cooling of a busbar tutorial model.

Run the Study: Nonisothermal Flow

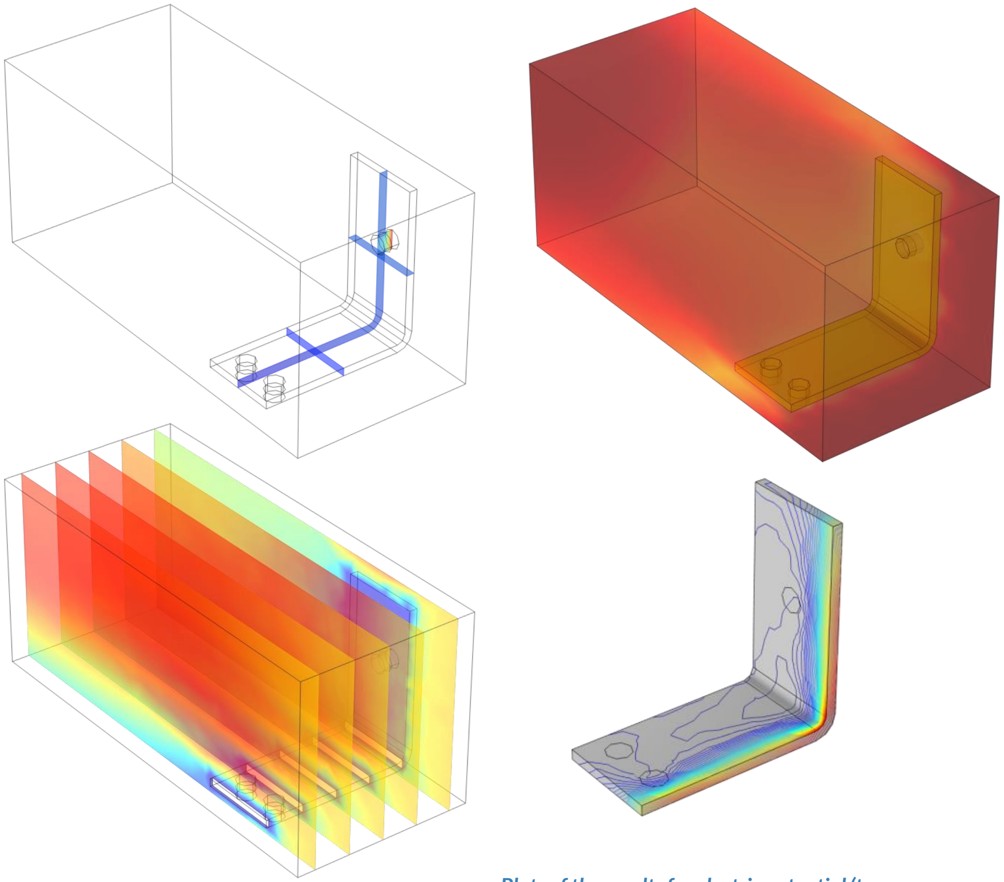
- Add a *Stationary* study
  - *Study 3*
- Change label for *Study 3* node to *Convective Cooling Analysis*
- Compute the model



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

Postprocess Results: Nonisothermal Flow

- Default plots generated by the software
  - Electric Potential
  - Temperature
  - Velocity
  - Pressure
- Create a plot of the current density
  - Duplicate *Current Density 2* plot and rename it *Current Density 3*
  - Change the *Dataset* to *Study 3/Solution 3 (sol 3)*



Plots of the results for electric potential (top left), temperature (top right), velocity (bottom left), and pressure (bottom right)

COMSOL