Modeling Exercise

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

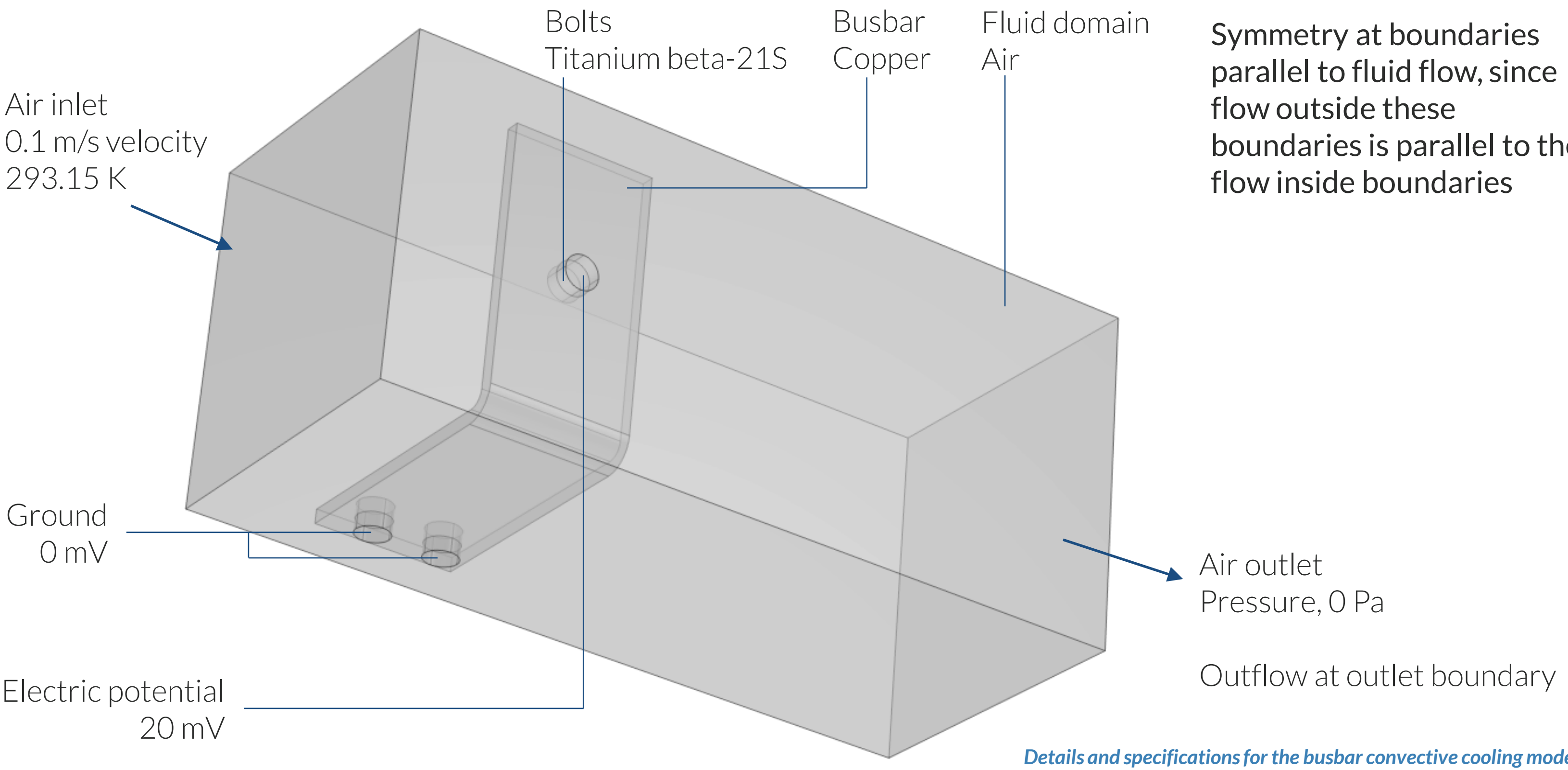
Introduction

- This modeling exercise demonstrates the concept of multiphysics modeling in COMSOL Multiphysics®
- Define the physics for the model using the manual approach with user-defined couplings
  - Note: This approach can be implemented different ways
    - This exercise demonstrates one of these such ways
  - Add and define the physics settings for the *Electric Currents* interface, followed by the *Heat Transfer in Solids* interface, followed by the *Laminar Flow (spf)* interface, and then manually couple the physics to simulate the resistive heating using a *Heat Source* domain feature and the convective cooling using the dependent variables of the fluid flow and heat transfer physics interfaces as inputs to each other
    - 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
  - 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

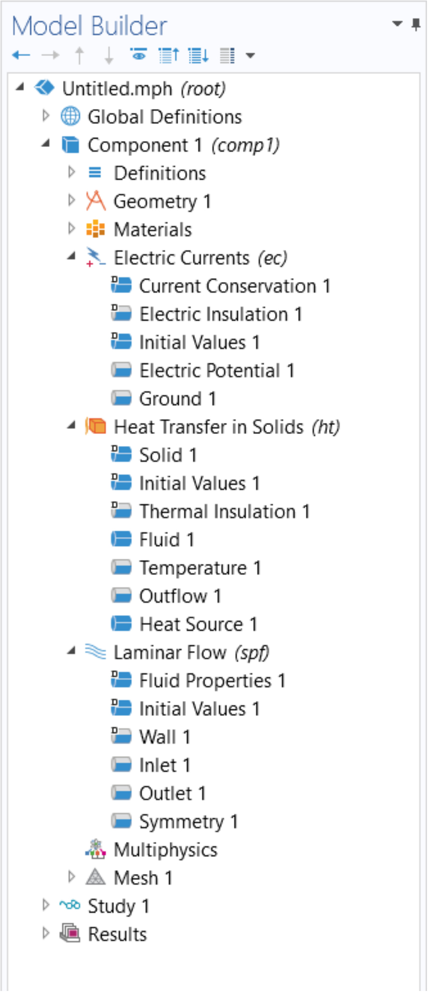
Details and specifications for the busbar convective cooling model

Manual 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 after the physics interfaces and the features to enable coupling the physis have been implemented

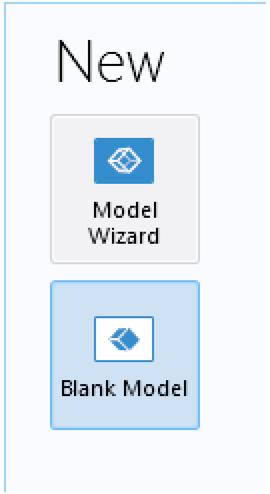
Modeling Workflow

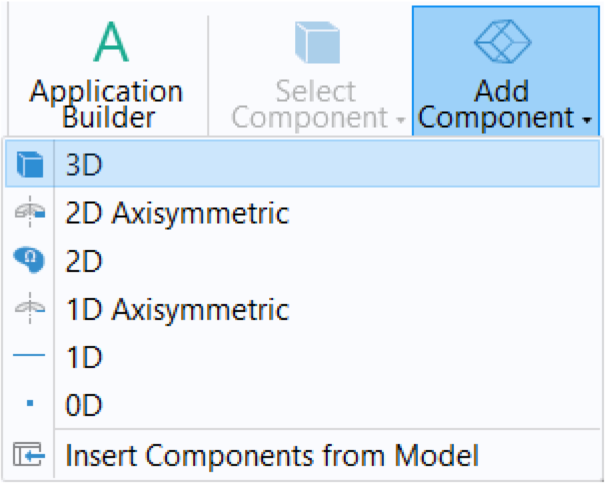
An outline of the steps used to set up, build, and compute this model to complete this modeling exercise is provided here:

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
   - Add *Laminar Flow (spf)* interface
   - Implement user-defined multiphysics couplings
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

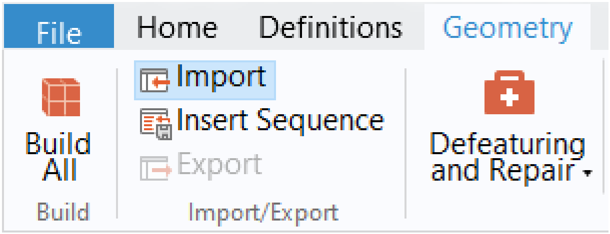


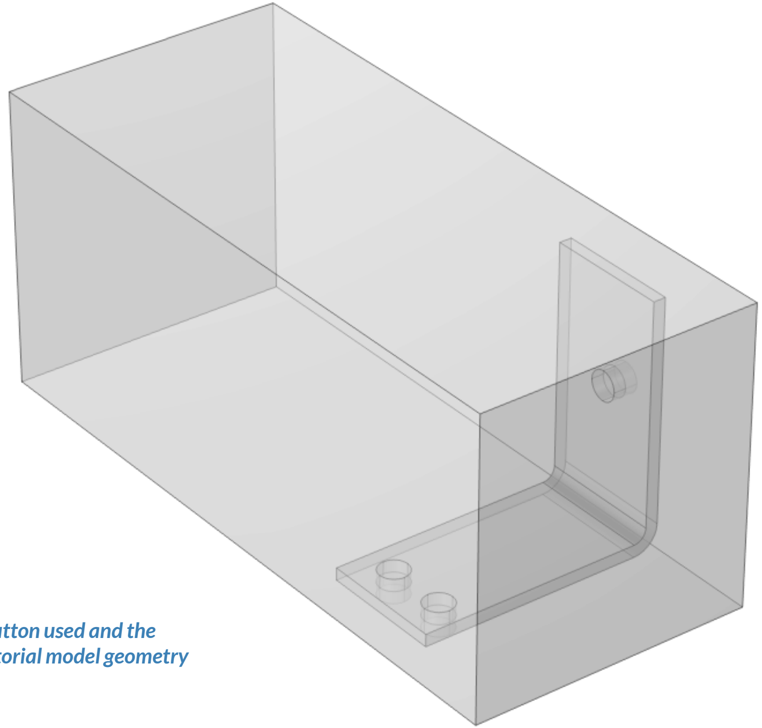


Screenshots of the steps performed to set up the model

Import Geometry

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

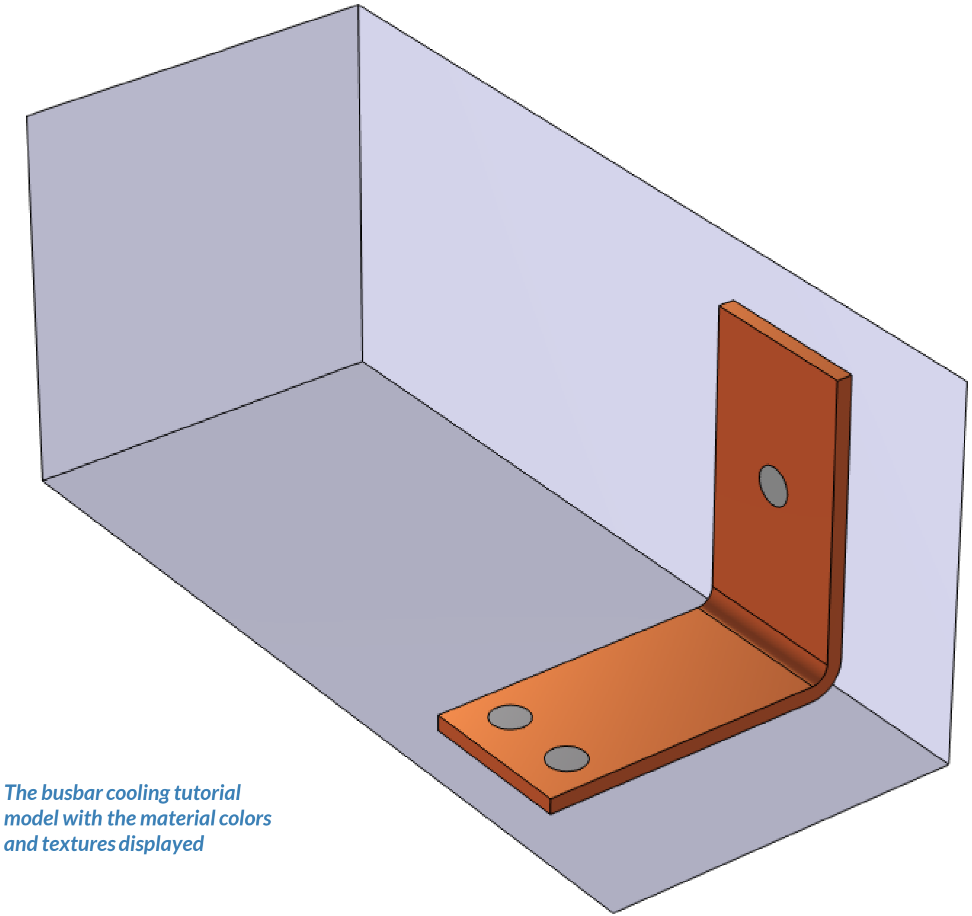




The Import button used and the busbar box tutorial model geometry

Assign Materials

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



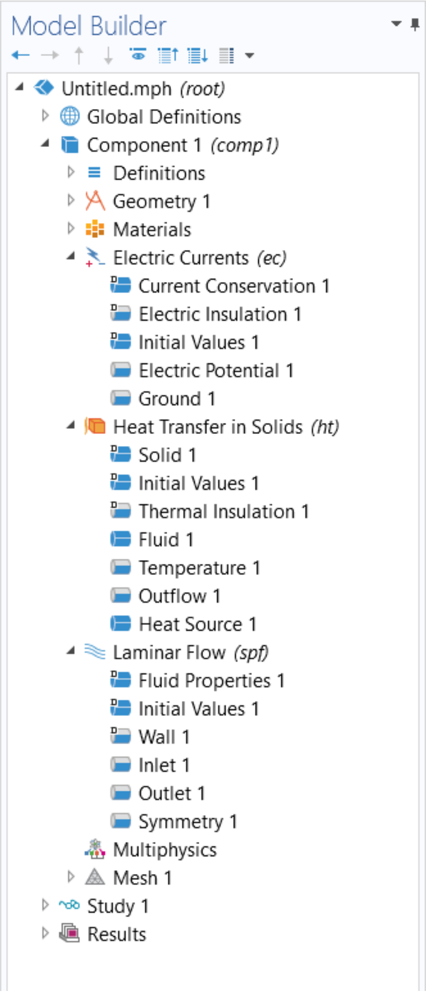
The busbar cooling tutorial model with the material colors and textures displayed

Manual 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*
   - *Laminar Flow (spf)*
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
   - *Laminar Flow (spf)* > *Fluid Properties* node
   - *Heat Transfer in Solids* > *Fluid* node



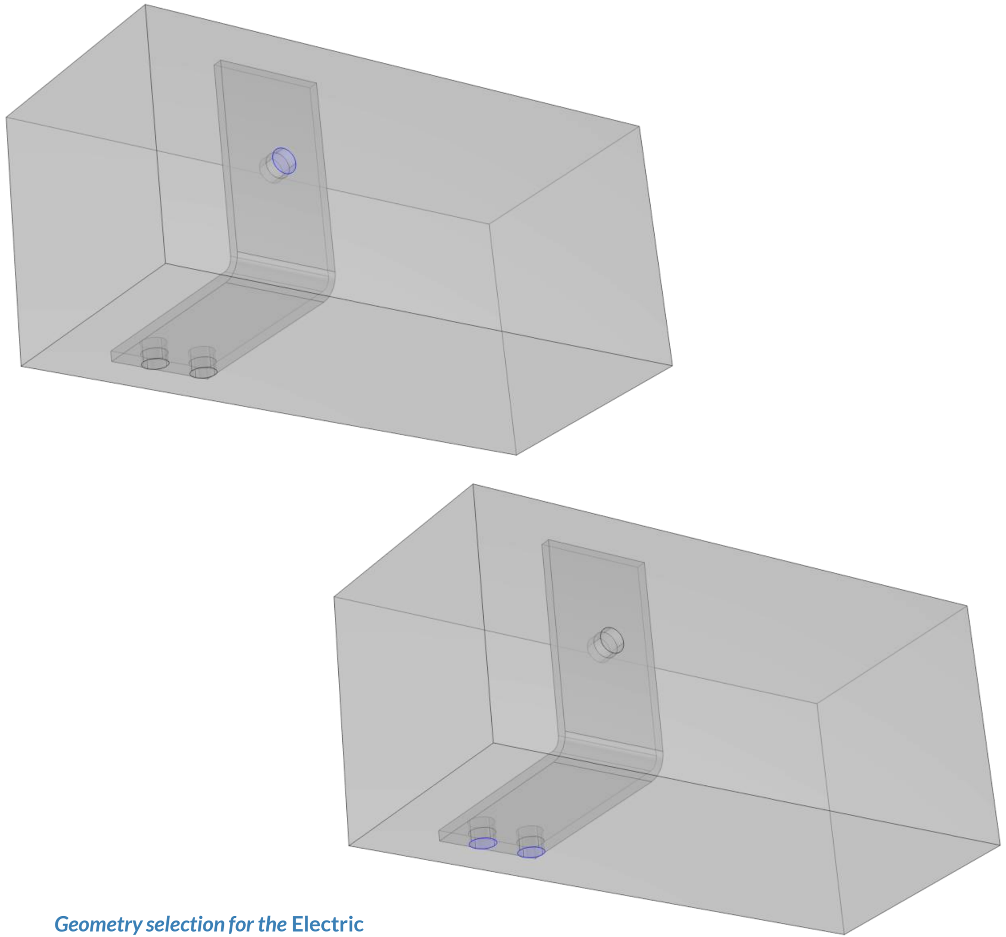
The model tree after the physics interfaces and the features to enable coupling the physics have been implemented

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

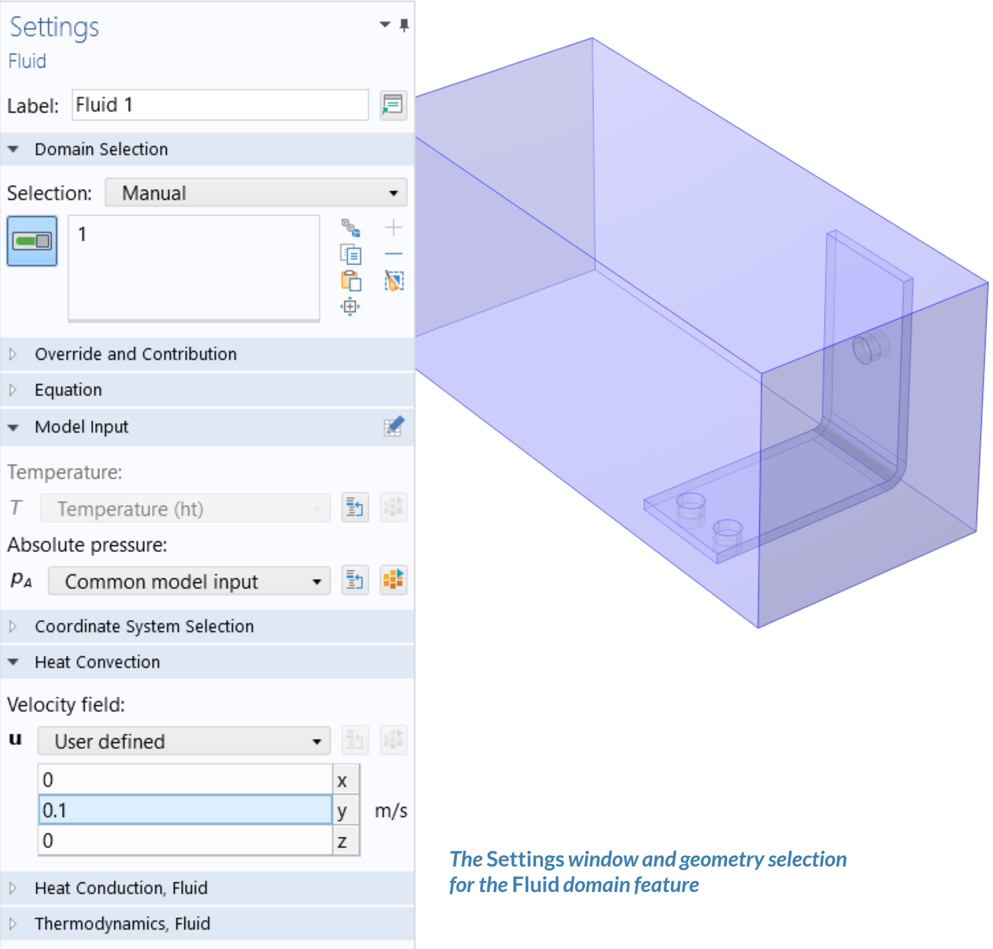


Geometry selection for the Electric Potential *(top) and* Ground *(bottom) boundary conditions*

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



The Settings window and geometry selection for the Fluid domain feature

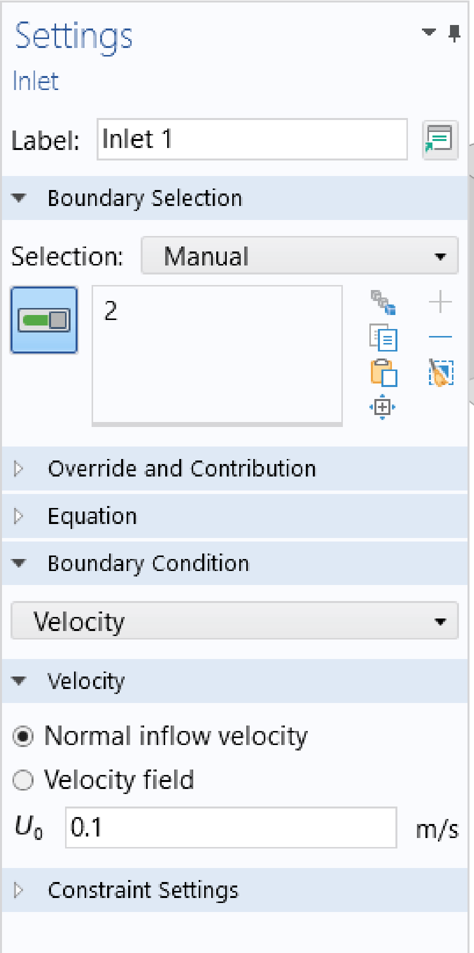
* = Refer to model specifications for values

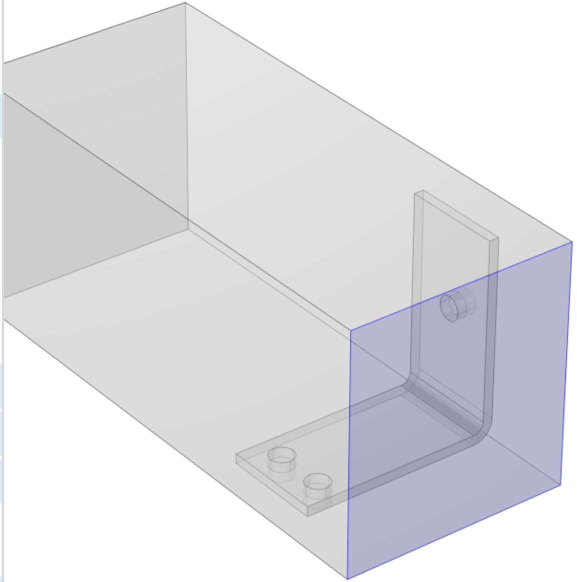
PHYSICS SETTINGS

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



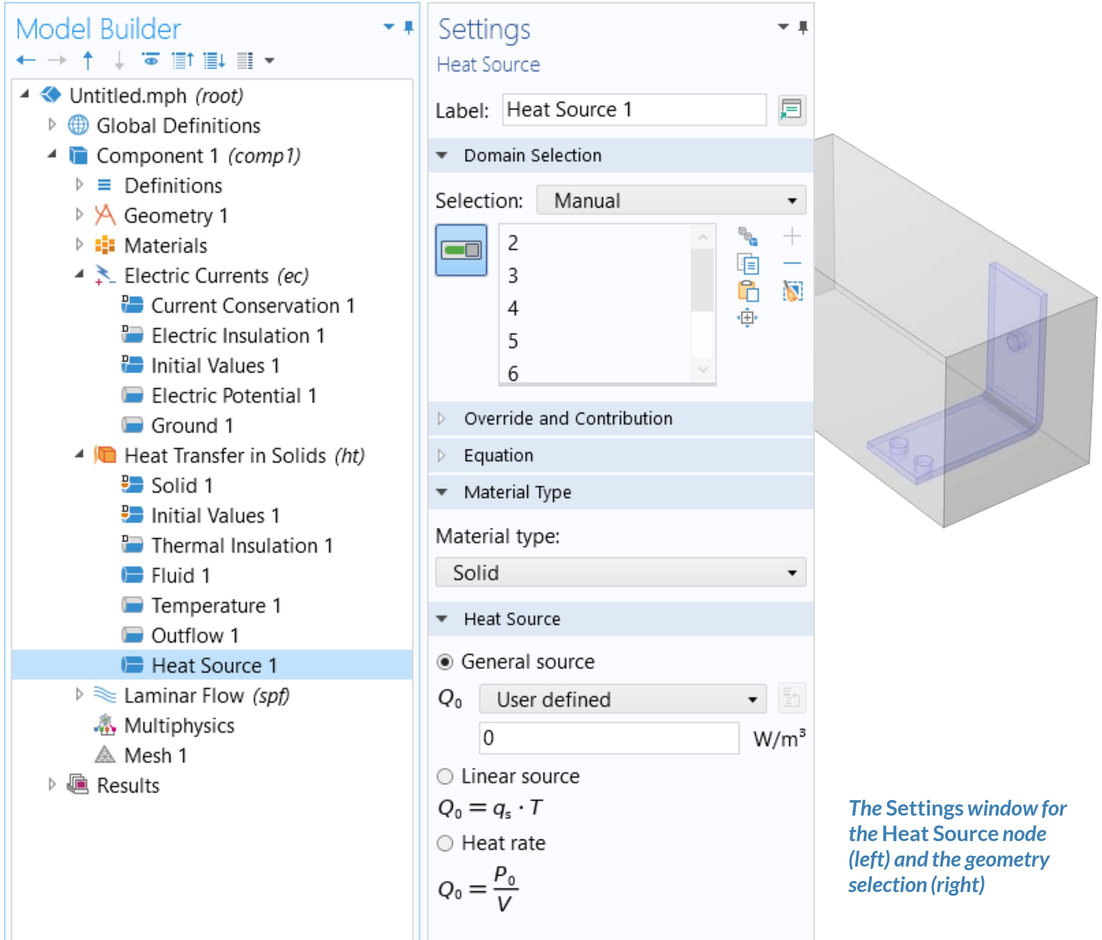


The Settings window and geometry selection for the Inlet *boundary condition*

MULTIPHYSICS SETTINGS

Electromagnetic Heating

- Add *Heat Source* domain feature
  - Defines heat generation within the domain
  - Used to map resistive losses as a heat source
- Update settings for *Heat Source* node
  - Check that the *Domain Selection* includes the busbar and bolt domains only
  - Choose *General source*
  - Enter expression that describes the appropriate quantity
    - Electric losses

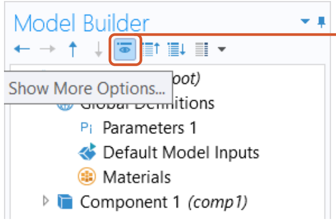


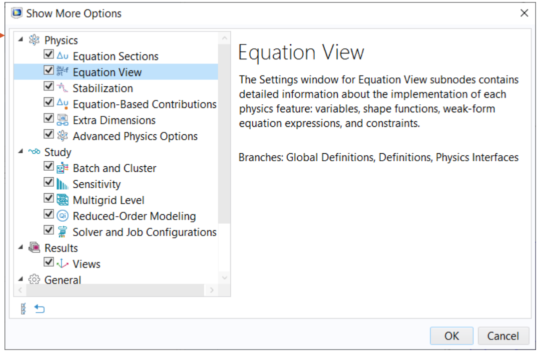
The Settings window for the Heat Source node (left) and the geometry selection (right)

MULTIPHYSICS SETTINGS

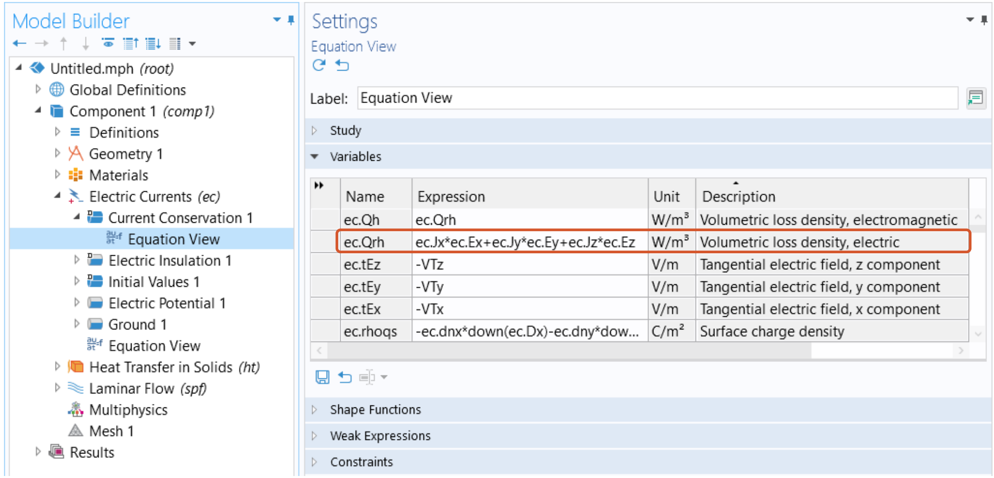
Electromagnetic Heating

- 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.*



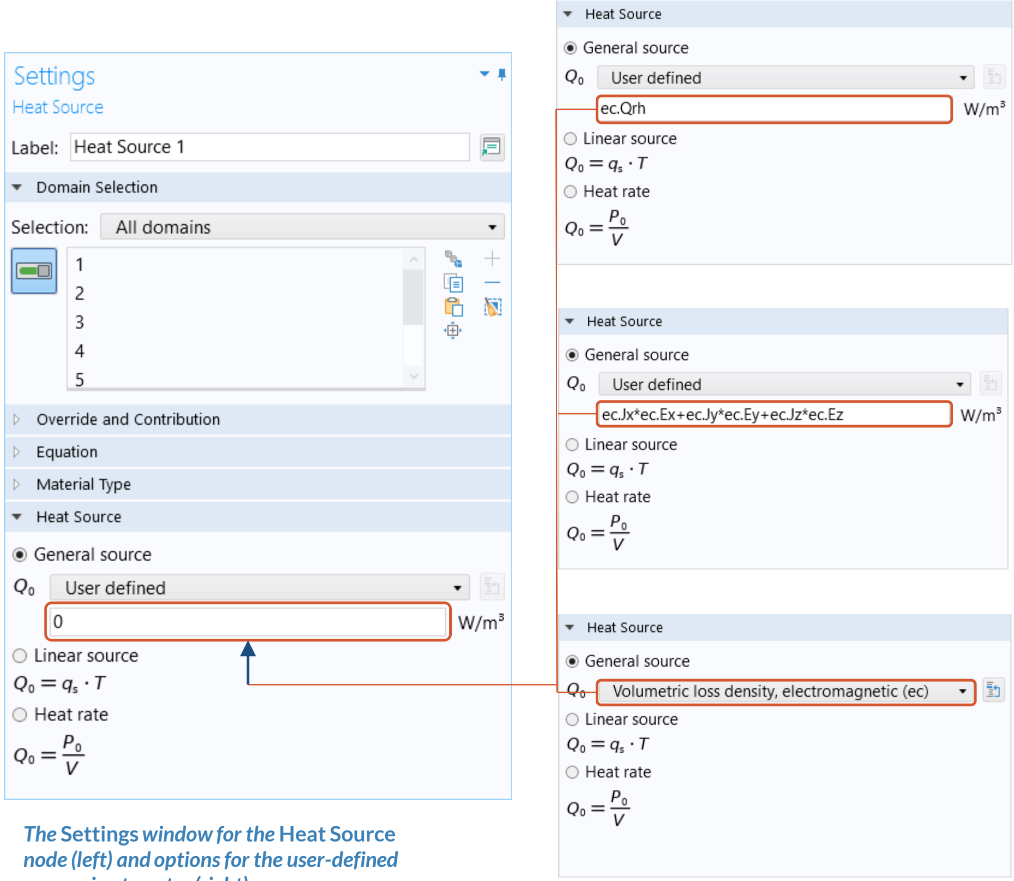
Electromagnetic Heating

- 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, 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)

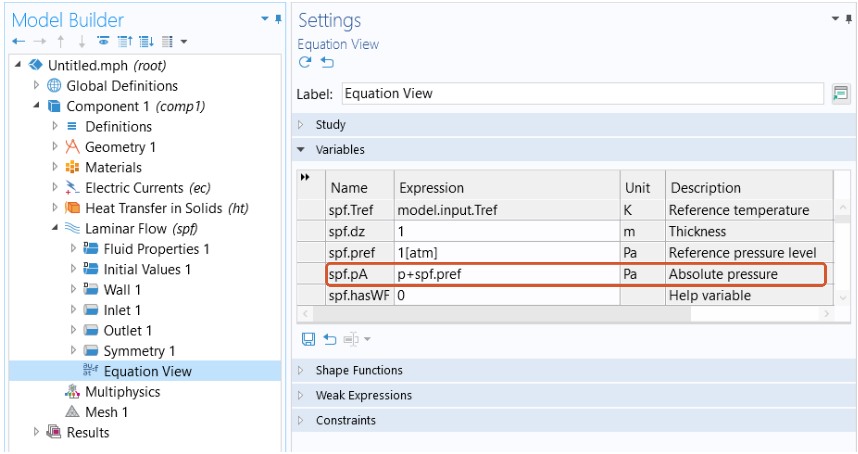


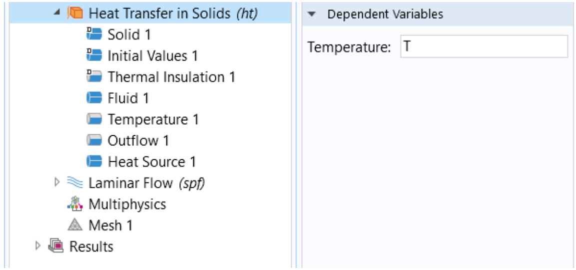
The Settings window for the Heat Source node (left) and options for the user-defined expression to enter (right)

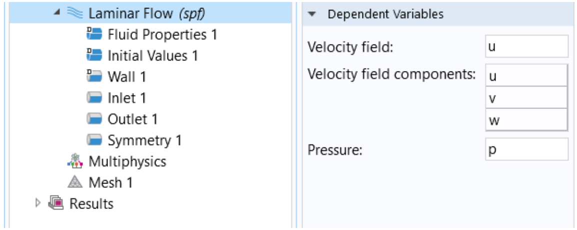
MULTIPHYSICS SETTINGS

Nonisothermal Flow

- *Heat Transfer in Solids* interface
  - Include absolute pressure from fluid flow interface as input
  - Include velocity field from fluid flow interface as velocity field for convective heat transfer
- *Laminar Flow (spf)* interface
  - Include temperature from heat transfer interface as input
    - Fluid properties depend on temperature







Equation View *node for the* Laminar Flow (spf) *interface (top) and the dependent variables for the* Laminar Flow (spf) *interface (middle) and* Heat Transfer in Solids *interface (bottom)*

MULTIPHYSICS SETUP

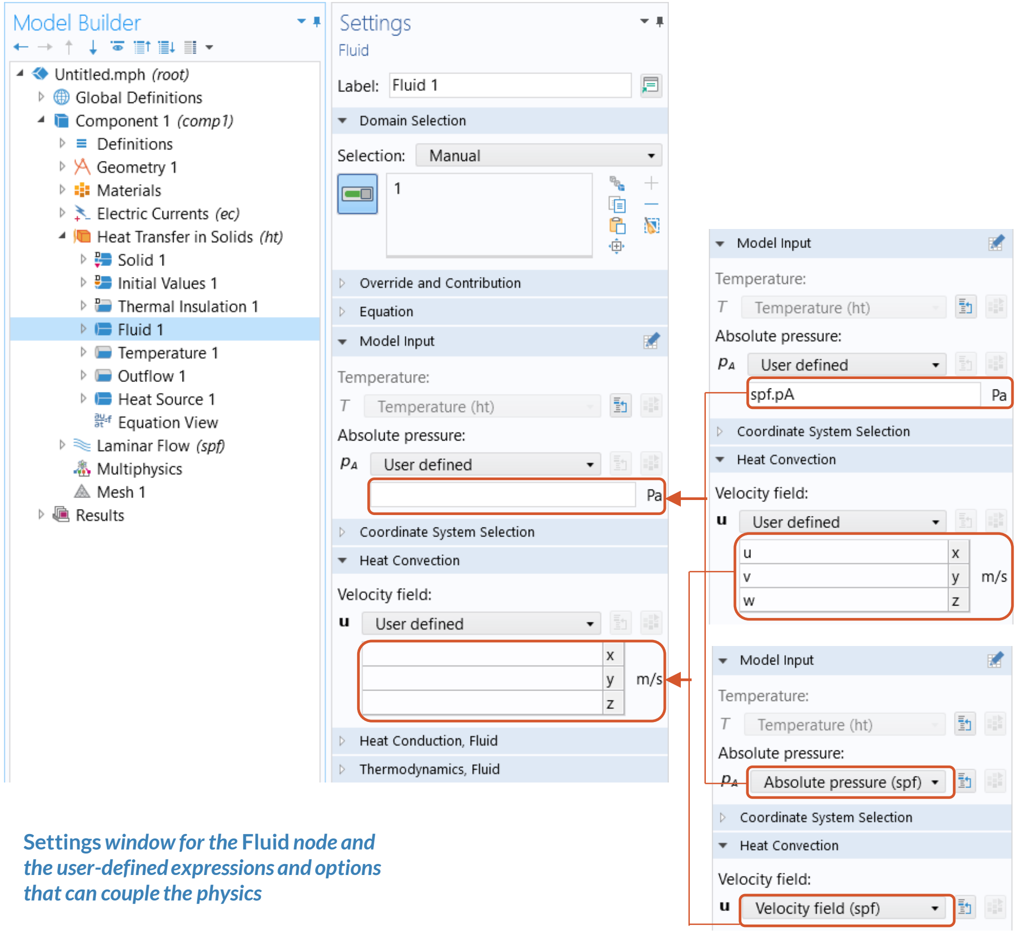
Nonisothermal Flow

Heat Transfer in Solids interface

1. Select *Fluid* node

2. Change the model inputs for the absolute pressure and velocity field to *User defined*

3. For the expression enter the absolute pressure and velocity field components from the *Laminar Flow (spf)* interface
   - Absolute pressure is available as a predefined physics variable
   - Alternatively select *Absolute pressure (spf)* and *Velocity field (spf)* from the respective drop-down menus



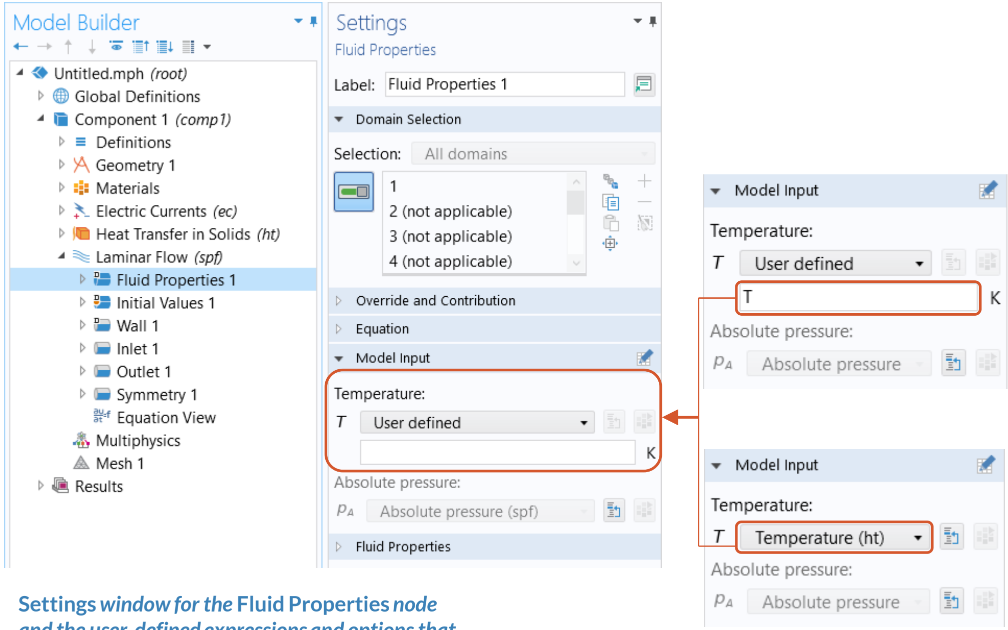
Settings window for the Fluid *node and the user-defined expressions and options that can couple the physics*

MULTIPHYSICS SETUP

Nonisothermal Flow

Laminar Flow (spf) interface

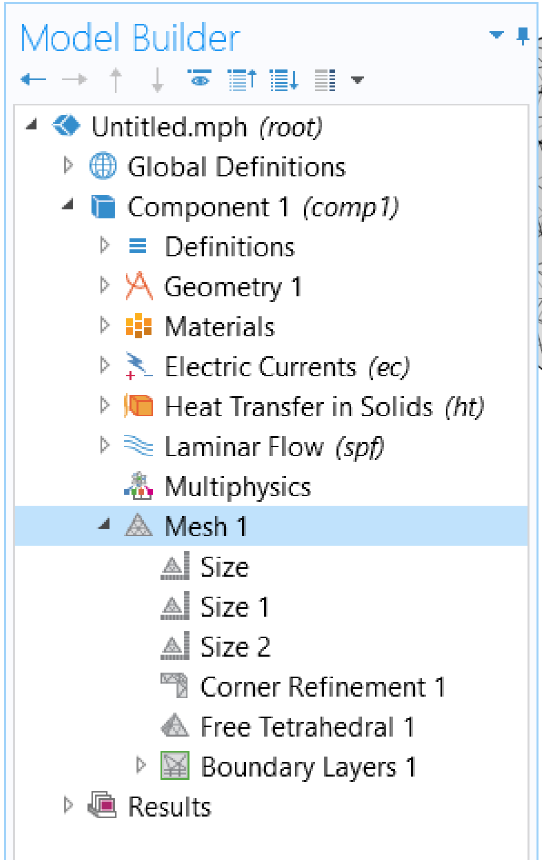
1. Select *Fluid Properties* node
2. Change the model input for temperature to *User defined*
3. For the expression, enter the temperature field from the *Heat Transfer in Solids* interface
   - Alternatively select *Temperature (ht)* from the *Temperature* model input drop-down menu

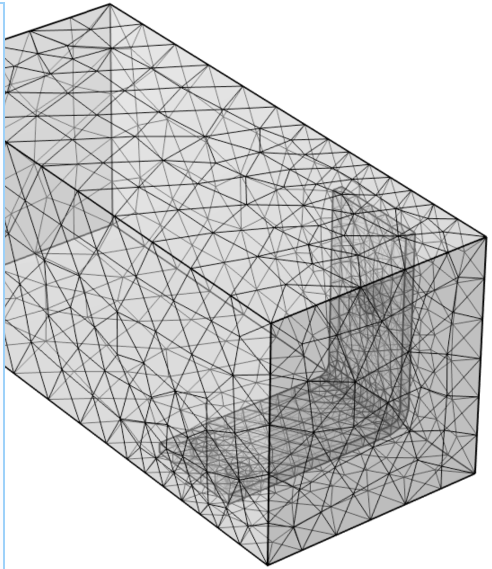


Settings window for the Fluid Properties node and the user-defined expressions and options that can couple the physics

Build the Mesh

- Build and then edit the mesh to reduce the computational resources required when recomputing 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

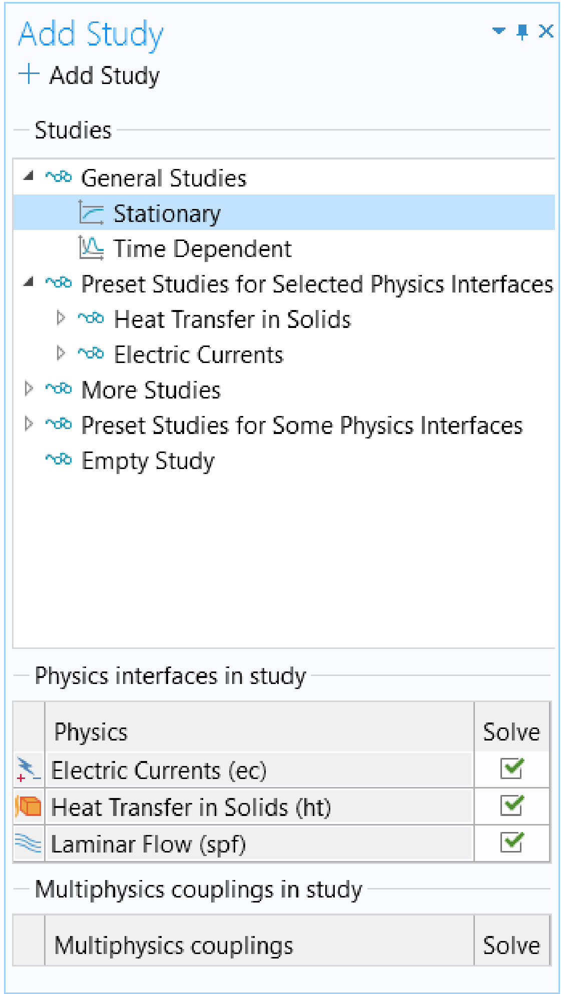




The meshing sequence and resultant mesh generated for the convective cooling of a busbar tutorial model

Run the Study

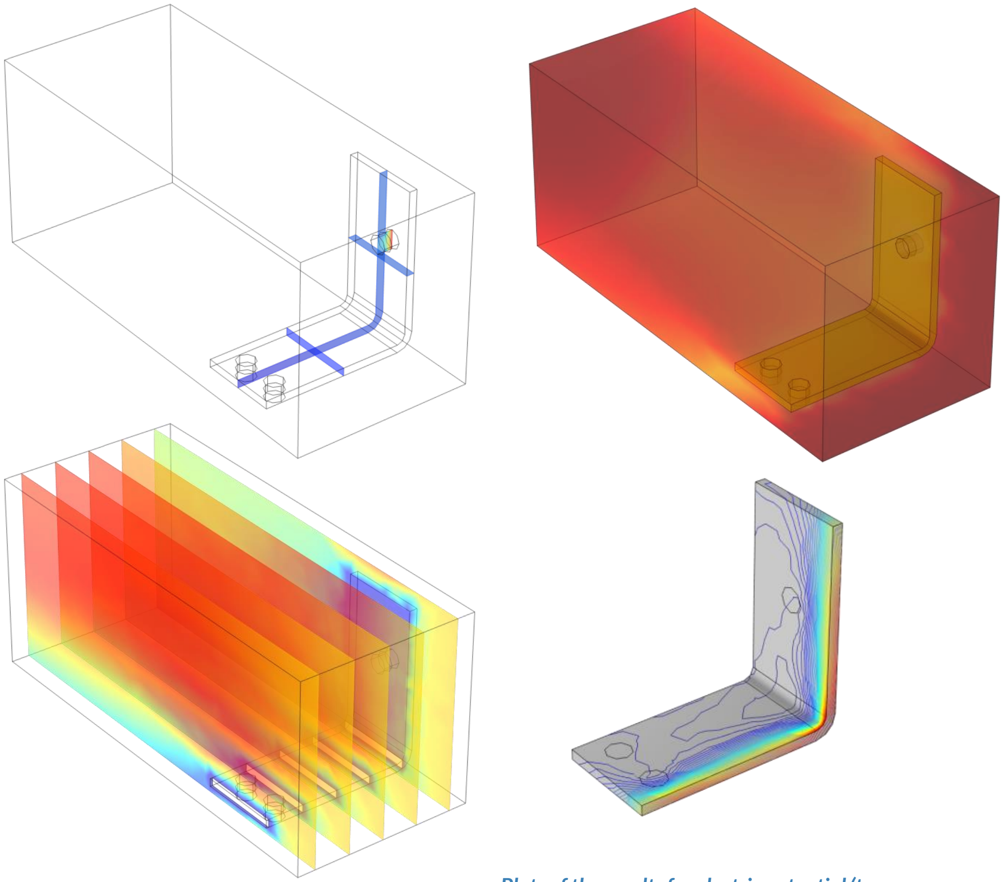
- Add a *Stationary* study
- Compute the model



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
  - Velocity
  - Pressure
- Create a plot of the current density
  - Add a *3D Plot Group*, rename it *Current Density*
  - Add a *Surface* plot
  - Use an expression that defines the current density norm
  - Use a *Manual Color Range*
    - *Minimum = 0*
    - *Maximum = 1e6*



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