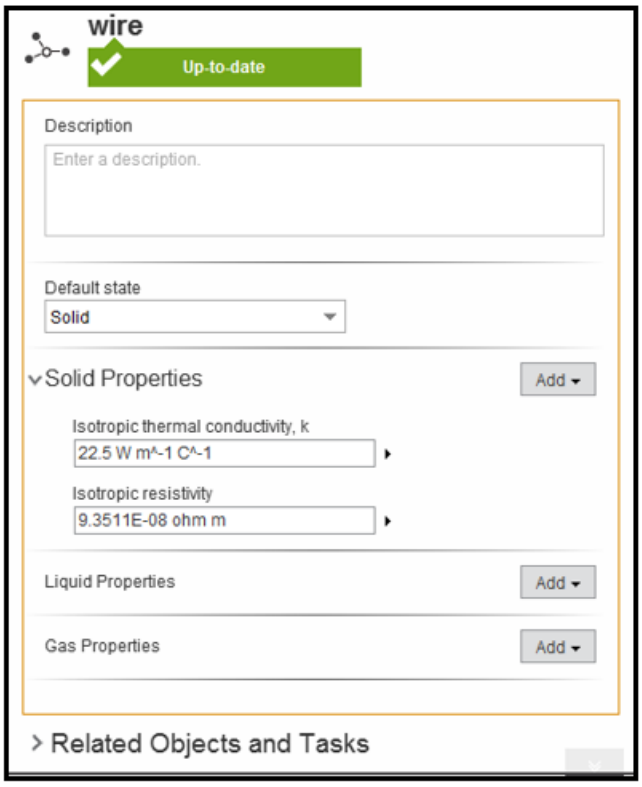
ANSYS AIM Thermal Analysis of an Electrical Wire - Physics Setup

Author(s): Joshua Wallace & Steve Scampoli, ANSYS Inc.

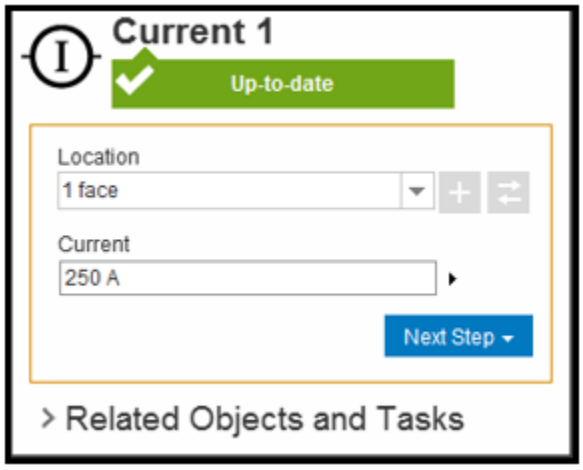
Physics Setup

Press the **Physics** tab in the workflow. Create a new material assignment by clicking on **Material Assignments**. Under **Material**, click on the drop down menu and create a new material called 'wire'. Then click on **wire** in order to provide material constants. Under **Default State** select **Solid** then **Add** > **Isotropic Thermal Conductivity** and value below. Follow the same process for adding **Isotropic Resistivity**.

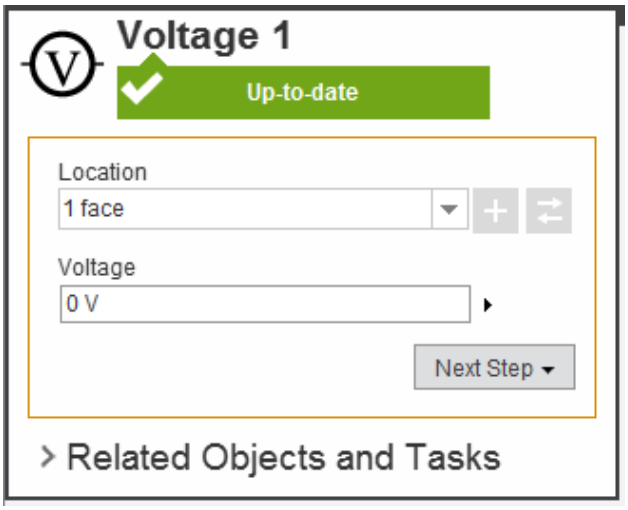


Boundary Conditions / Forces

Next, the boundary conditions for the wire need to be created. Right click > **Add** > **Electromagnetic Conditions**> **Current**. Select on end of the wire and input a **Current** of 250 amps. Since the model of the wire is a one quarter symmetric model, one quarter of the total current will be entered for the current value.



Right click again > **Electromagnetic Conditions** > **Voltage**. Select the other end of the wire as the **Location** and input 0 V as the **Voltage**. The voltage condition allows the current to exit the current conduction model.



Right click again > **Solid Thermal Conditions** > **Convection**. Select the curved face as the location and input the **Heat Transfer Coefficient** and **Convect ion Temperature** as shown below in the image.

No boundary conditions are required on the symmetry surfaces as a symmetry boundary condition for electric conduction and heat transfer is a natural (unspecified) boundary condition. Now that the Boundary conditions are set, the results are ready to be calculated.

Go to Step 4: Numerical Solution/Results

Go to all ANSYS AIM Learning Modules