

ANSYS AIM Thermal Analysis of an Electrical Wire - Physics Setup

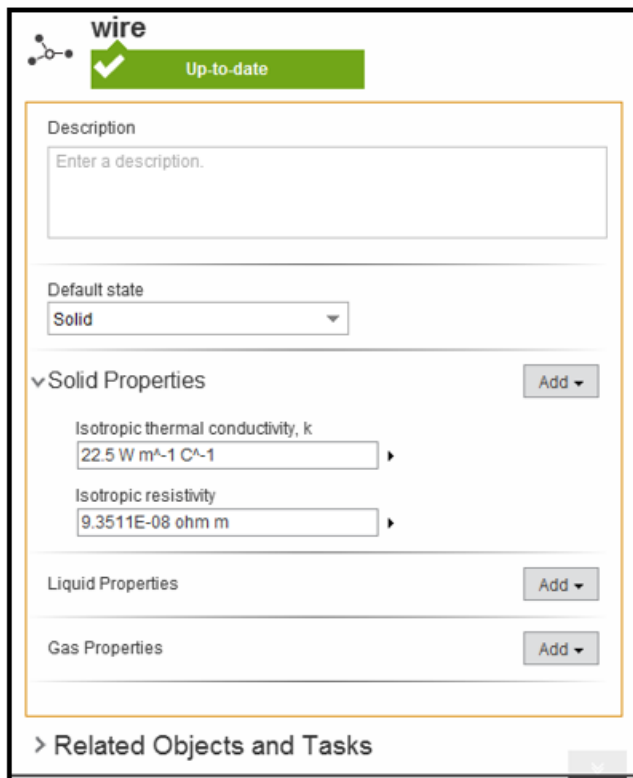
Author(s): Joshua Wallace & Steve Scampoli, ANSYS Inc.

[Problem Specification](#)

1. Pre-Analysis & Start-Up
2. Geometry
3. Physics Setup
4. Numerical Solution/Results
5. Verification & Validation

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**.




The screenshot shows the 'wire' material properties dialog box in ANSYS AIM. At the top, there is a green status bar with a checkmark and the text 'Up-to-date'. Below this is a 'Description' field with the placeholder text 'Enter a description.'. Underneath is a 'Default state' dropdown menu currently set to 'Solid'. The 'Solid Properties' section is expanded, showing two input fields: 'Isotropic thermal conductivity, k' with the value '22.5 W m^-1 C^-1' and 'Isotropic resistivity' with the value '9.3511E-08 ohm m'. Each of these sections has an 'Add' button with a dropdown arrow. Below the solid properties are sections for 'Liquid Properties' and 'Gas Properties', each also with an 'Add' button. At the bottom of the dialog is a section labeled '> Related Objects and Tasks'.

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.



Current 1


Up-to-date

Location

+


↔

Current


Next Step ▾

> Related Objects and Tasks

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.



Voltage 1


Up-to-date

Location

+


↔

Voltage


Next Step ▾

> Related Objects and Tasks

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



Convection 1


Up-to-date

Location

+

↔

Heat transfer coefficient

Convection temperature

Next Step ▾

> Related Objects and Tasks

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](#)