

# ANSYS AIM - Thermal Analysis of an Electrical Wire

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

## Thermal Analysis of an Electrical Wire

Created using ANSYS AIM 18.2

### Problem Specification

For this demonstration we are asked to determine the centerline temperature,  $T_c$ , and surface temperature of a bare steel wire carrying a current,  $I$ , and having a resistance,  $R$ . The surface convection coefficient between the wire and the ambient air (at temperature  $T_a$ ) is  $h$ .

Given values:

$$k = 13 \text{ BTU/hr} \cdot \text{ft} \cdot \text{F}$$

$$h = 5 \text{ BTU/hr} \cdot \text{ft}^2 \cdot \text{F}$$

$$R = 0.0001 \text{ } \Omega/\text{ft}$$

$$T_a = 70 \text{ F}$$

$$I = 1000 \text{ A}$$

[Go to Step 1: Pre-Analysis & Start-Up](#)

[Go to all ANSYS AIM Learning Modules](#)