

Inductive Heating of a Copper Cylinder

Introduction

The induced currents in a copper cylinder produce heat, and when the temperature rises, the electric conductivity of the copper changes. Solving the heat transfer problem simultaneously with the field propagation problem is therefore crucial for an accurate description of this process.

The heating caused by the induced currents is called inductive heating. Generally, heating due to currents is also called resistive heating or ohmic heating.

A challenge in induction heating is that the high current in the induction coils requires active cooling. This can be obtained by making the coil conductors hollow and circulating water inside. Even for rather modest flow rates, the coolant flow becomes highly turbulent which makes the heat transfer between conductor and fluid very efficient. This example illustrates a simplified way of modeling water cooling based on the assumption of turbulent flow and instantaneous mixing.

For mechanical support and electrical insulation, the cylinder and coil are embedded in FR4 composite material.

Model Definition

The system to be solved is given by

$$j\omega\sigma(T)\mathbf{A} + \nabla \times (\mu^{-1}\nabla \times \mathbf{A}) = 0$$
$$\rho C_p \frac{\partial T}{\partial t} - \nabla \cdot k \nabla T = Q(T, \mathbf{A})$$

where ρ is the density, C_p is the specific heat capacity, k is the thermal conductivity, and Q is the inductive heating. Note that in 2D axisymmetric when solving the out-of-plane vector potential (as in this model), the **Magnetic Fields** interface uses *The Covariant Formulation* described in the *AC/DC Module User's Guide*, where the dependent variable is $\Psi = rA_{\phi}$, instead of A_{ϕ} .

The electric conductivity of copper, σ , is given by the expression

$$\sigma = \frac{1}{[\rho_0(1 + \alpha(T - T_0))]}$$

where ρ_0 is the resistivity at the reference temperature $T_0 = 293$ K, α is the temperature coefficient of the resistivity, and *T* is the actual temperature in the domain.

The time average of the inductive heating over one period, is given by

$$Q = \frac{1}{2}\sigma |\mathbf{E}|^2$$

The coil conductor is cooled by a turbulent water flow in an internal cooling channel. This is emulated by a combination of a high effective thermal conductivity and a homogenized out-of-plane convective loss term:

$$Q_c = \frac{\frac{dM}{dt}C_p(T_{in} - T)}{2\pi r A_c}$$

where $\frac{dM}{dt}$ is the water mass flow, T_{in} is the water inlet temperature, r is the radial coordinate, and A_c is the cross-section area of the cooling channel.

Results and Discussion

The temperature after 10 h is shown in Figure 1. The average temperature of the copper cylinder has increased from 293 K to 346 K during this time, and the current in the coil has an amplitude of 2 kA.



Figure 1: Temperature distribution after 10 h.



Figure 2: The plot shows the temperature evolution in the center of the copper cylinder and in the cooling channel.

Application Library path: ACDC_Module/Electromagnetic_Heating/

inductive_heating

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click 🔗 Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 🖚 2D Axisymmetric.
- 2 In the Select Physics tree, select Heat Transfer>Electromagnetic Heating> Induction Heating.

- 3 Click Add.
- 4 Click 🔿 Study.
- 5 In the Select Study tree, select Preset Studies for Selected Multiphysics>Frequency-Transient.
- 6 Click M Done.

GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
10	2e3[A]	2000 A	Current
Т0	293[K]	293 K	Reference temperature
r0	1.754e-8[ohm*m]	I. 754E-8 Ω·m	Resistivity at T=TO
al	0.0039[1/K]	0.0039 I/K	Temperature coefficient
Rc	5[mm]	0.005 m	Cooling channel radius
Ac	pi*Rc^2	7.854E-5 m ²	Cooling channel x- section
Mt	1[kg/min]	0.016667 kg/s	Cooling water mass flow rate
Tin	10[degC]	283.15 K	Cooling water inlet temperature

GEOMETRY I

Rectangle 1 (r1)

- I In the Geometry toolbar, click Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type **0.2**.
- 4 In the **Height** text field, type 0.5.
- 5 Locate the **Position** section. In the **z** text field, type -0.25.
- 6 Click 틤 Build Selected.

Rectangle 2 (r2)

- I In the Geometry toolbar, click Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type **0.03**.
- 4 In the **Height** text field, type 0.1.
- 5 Locate the **Position** section. In the z text field, type -0.05.
- 6 Click 틤 Build Selected.

Circle I (c1)

- I In the **Geometry** toolbar, click \bigcirc **Circle**.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type 0.01.
- 4 Locate the **Position** section. In the **r** text field, type 0.05.
- 5 Click 📄 Build Selected.

Circle 2 (c2)

- I Right-click Circle I (cl) and choose Duplicate.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type Rc.
- 4 Click 틤 Build Selected.

Form Union (fin)

- I In the Model Builder window, click Form Union (fin).
- 2 In the Settings window for Form Union/Assembly, click 틤 Build Selected.

ADD MATERIAL

- I In the Home toolbar, click 🙀 Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-in>FR4 (Circuit Board).
- 4 Click Add to Component in the window toolbar.
- 5 In the tree, select **AC/DC>Copper**.
- 6 Click Add to Component in the window toolbar.
- 7 In the tree, select **Built-in>Water**, liquid.
- 8 Click Add to Component in the window toolbar.
- 9 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

MATERIALS

Copper (mat2)

- I In the Model Builder window, under Component I (compl)>Materials click Copper (mat2).
- 2 Select Domains 2 and 3 only.
- **3** In the **Model Builder** window, expand the **Copper (mat2)** node, then click **Linearized resistivity (ltr)**.
- 4 In the Settings window for Linearized Resistivity, locate the Output Properties section.
- **5** In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Reference resistivity	rho0	r0	Ω·m	IxI
Resistivity temperature coefficient	alpha	al	I/K	IxI
Reference temperature	Tref	то	К	IxI

Water, liquid (mat3)

- I In the Model Builder window, click Water, liquid (mat3).
- **2** Select Domain 4 only.

The built-in water material does not provide the electric permittivity and the magnetic permeability. Add those values.

- 3 In the Settings window for Material, locate the Material Contents section.
- **4** In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	80	I	Basic

Increase the thermal conductivity of the water to model the efficient heat transport in turbulent flow.

5 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	1e3	W/(m·K)	Basic

MAGNETIC FIELDS (MF)

Add a separate **Ampère's Law** feature in the copper regions to specify a temperaturedependent resistivity.

Ampère's Law 2

- I In the Model Builder window, under Component I (compl) right-click Magnetic Fields (mf) and choose Ampère's Law.
- **2** Select Domains 2 and 3 only.
- 3 In the Settings window for Ampère's Law, locate the Constitutive Relation Jc-E section.
- 4 From the Conduction model list, choose Linearized resistivity.

Coil I

- I In the Physics toolbar, click 🔵 Domains and choose Coil.
- **2** Select Domain 3 only.
- 3 In the Settings window for Coil, locate the Coil section.
- **4** In the I_{coil} text field, type I0.

HEAT TRANSFER IN SOLIDS (HT)

Set up the Heat Transfer boundary conditions.

I In the Model Builder window, under Component I (comp1) click Heat Transfer in Solids (ht).

Temperature 1

- I In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
- **2** Select Boundaries 2, 7, and 9 only.
- 3 In the Settings window for Temperature, locate the Temperature section.
- **4** In the T_0 text field, type T0.

Heat Source 1

- I In the Physics toolbar, click **Domains** and choose **Heat Source**.
- **2** Select Domain 4 only.

3 In the Settings window for Heat Source, locate the Heat Source section.

4 In the Q_0 text field, type Mt*ht.Cp*(Tin-T)/(2*pi*r*Ac).

MESH I

In the Model Builder window, under Component I (comp1) right-click Mesh I and choose Build All.

STUDY I

Step 1: Frequency-Transient

- I In the Model Builder window, under Study I click Step I: Frequency-Transient.
- 2 In the Settings window for Frequency-Transient, locate the Study Settings section.
- 3 From the Time unit list, choose h.
- 4 Click Range.
- 5 In the Range dialog box, type 10[min] in the Step text field.
- 6 In the **Stop** text field, type 10[h].
- 7 Click Replace.
- 8 In the Settings window for Frequency-Transient, locate the Study Settings section.
- 9 In the Frequency text field, type 500[Hz].

IO In the **Home** toolbar, click **= Compute**.

RESULTS

Temperature, 3D (ht)

The revolution plot shows the temperature distribution after 10 hours; compare with Figure 1.

Create point datasets for plotting the temperature evolution in the copper cylinder and in the cooling channel.

Cut Point 2D 1

- I In the Model Builder window, expand the Results>Datasets node.
- 2 Right-click Results>Datasets and choose Cut Point 2D.
- 3 In the Settings window for Cut Point 2D, locate the Point Data section.
- **4** In the **r** text field, type **0**.
- **5** In the **z** text field, type **0**.

Cut Point 2D 2

- I Right-click Cut Point 2D I and choose Duplicate.
- 2 In the Settings window for Cut Point 2D, locate the Point Data section.
- **3** In the **r** text field, type **0.05**.

ID Plot Group 5

- I In the **Results** toolbar, click \sim **ID Plot Group**.
- 2 In the Settings window for ID Plot Group, locate the Legend section.
- 3 From the Position list, choose Middle right.

Point Graph 1

- I Right-click ID Plot Group 5 and choose Point Graph.
- 2 In the Settings window for Point Graph, locate the Data section.
- 3 From the Dataset list, choose Cut Point 2D I.
- 4 From the Solution parameters list, choose From parent.
- 5 Click Replace Expression in the upper-right corner of the y-Axis Data section. From the menu, choose Component I (compl)>Heat Transfer in Solids>Temperature>T Temperature K.
- 6 Click to expand the Legends section. Select the Show legends check box.
- 7 From the Legends list, choose Manual.
- 8 In the table, enter the following settings:

Legends

Cylinder center

Point Graph 2

- I Right-click Point Graph I and choose Duplicate.
- 2 In the Settings window for Point Graph, locate the Data section.
- 3 From the Dataset list, choose Cut Point 2D 2.
- **4** Locate the **Legends** section. In the table, enter the following settings:

Legends

Cooling channel

5 In the ID Plot Group 5 toolbar, click 💽 Plot.

The plot shows the temperature evolution in the center of the copper cylinder and in the cooling channel; compare with Figure 2.

Finish the modeling session by saving a representative model thumbnail.

Temperature, 3D (ht) Click the 4 Zoom Extents button in the Graphics toolbar.

ROOT

- I In the Model Builder window, click the root node.
- 2 In the root node's Settings window, locate the Presentation section.
- 3 Find the Thumbnail subsection. Click Set from Graphics Window.

12 | INDUCTIVE HEATING OF A COPPER CYLINDER