

Superconducting Wire

Current can flow in a superconducting wire with practically zero resistance, although factors including temperature, current density, and magnetic field can limit this phenomenon. This model solves a time-dependent problem of a current building up in a superconducting wire close to the critical current density. This model is based on a suggestion by Dr. Roberto Brambilla, CESI, Superconductivity Dept., Milano, Italy.

Introduction

The Dutch physicist Heike Kamerlingh Onnes discovered superconductivity in 1911. He cooled mercury to the temperature of liquid helium (4 K) and observed that its resistivity suddenly disappeared. Research in superconductivity reached a peak during the 1980s in terms of activity and discoveries, especially when scientists uncovered the superconductivity of ceramics. In particular, it was during this decade that researchers discovered YBCO—a ceramic superconductor composed of yttrium, barium, copper, and oxygen with a critical temperature above the temperature of liquid nitrogen. However, researchers have not yet created a room-temperature superconductor, so much work remains for the broad commercialization of this area.

This model illustrates how current builds up in a cross section of a superconducting wire; it also shows where critical currents produce a swelling in the non-superconducting region.

Model Definition

The dependence of resistivity on the amount of current makes it difficult to solve the problem using the standard physics interfaces in the AC/DC Module. The reason is this: a circular dependency arises because the current-density calculation contains the resistivity, leading to a resistivity that is dependent on itself.

An alternate approach uses the magnetic field as the dependent variable, and you can then calculate the current as

$$\mathbf{J} = \nabla \times \mathbf{H} . \quad (1)$$

The electric field is a function of the current, and Faraday's law determines the complete system as in

$$\nabla \times \mathbf{E}(\mathbf{J}) = -\mu \frac{\partial \mathbf{H}}{\partial t} \quad (2)$$

where $\mathbf{E}(\mathbf{J})$ is the current-dependent electric field. The model calculates this field with the empirical formula

$$\mathbf{E}(\mathbf{J}) = \begin{cases} \mathbf{0} & |\mathbf{J}| < J_C \\ E_0 \left(\frac{|\mathbf{J}| - J_C}{J_C} \right)^\alpha \frac{\mathbf{J}}{|\mathbf{J}|} & |\mathbf{J}| \geq J_C \end{cases} \quad (3)$$

where E_0 and α are constants determining the nonlinear behavior of the transition to zero resistivity, and J_C is the critical current density, which decreases as temperature increases.

For the superconductor YBCO, this model uses the following parameter values (Ref. 1):

PARAMETER	VALUE
E_0	0.0836168 V/m
α	1.449621256
J_C	17 MA
T_C	92 K

Systems with two curl operators are best dealt with using vector elements (edge elements). This is the default element for the physics interfaces in the AC/DC Module that solve similar equations. This particular formulation for the superconducting system is not available in the AC/DC Module, so you must define it using the General Form PDE interface. In addition, the model uses higher-order vector elements, called *curl elements* in COMSOL Multiphysics. The resulting system becomes

$$D_a \frac{d\mathbf{H}}{dt} + \nabla \cdot \boldsymbol{\Gamma} = \mathbf{F} \Rightarrow \begin{bmatrix} \mu_0 & 0 \\ 0 & \mu_0 \end{bmatrix} \cdot \begin{bmatrix} \frac{dH_x}{dt} \\ \frac{dH_y}{dt} \end{bmatrix} + \nabla \cdot \begin{bmatrix} 0 & E_z(J_z) \\ -E_z(J_z) & 0 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}. \quad (4)$$

For symmetry reasons, the current density has only a z -component.

The model controls current through the wire with its outer boundary condition. Because Ampère's law must hold around the wire, a line integral around it must add

up to the current through the wire. Cylindrical symmetry results in a known magnetic field at the outer boundary

$$\oint \mathbf{H} \cdot d\mathbf{l} = 2\pi r H_\phi = I_{\text{wire}} \Rightarrow H_\phi = \frac{I_{\text{wire}}}{2\pi r} . \quad (5)$$

For vector elements, the Dirichlet boundary conditions add a constraint on the tangential component of the vector field. If the field components are called H_x and H_y , the tangential counterparts are tH_x and tH_y .

Results and Discussion

The model applies a simple transient exponential function as the current through the wire, reaching a final value of 1 MA. This extremely large current is necessary if the superconducting wire is to reach its critical current density. Plotting the current density at different time instants shows the swelling of the region in which the current flows. This swelling comes from the transition out of the superconducting state at current densities exceeding J_C . [Figure 1](#) presents a plot of the current density at $t = 0.1$ s.

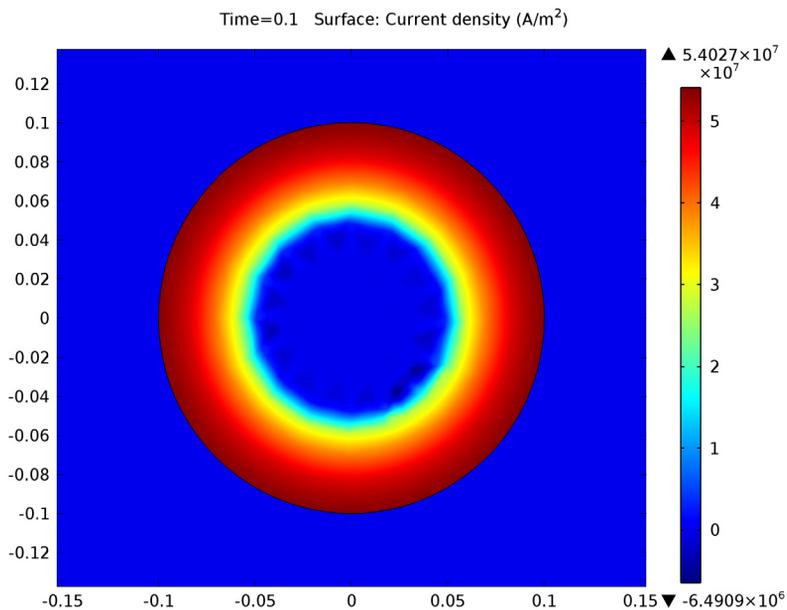


Figure 1: The current density at 0.1 s.

Reference

1. R. Pecher, M.D. McCulloch, S.J. Chapman, L. Prigozhin, and C.M. Elliotth, “3D-modelling of bulk type-II superconductors using unconstrained H-formulation,” *6th European Conf. Applied Superconductivity* (EUCAS 2003).

Model Library path: ACDC_Module/General_Industrial_Applications/
superconducting_wire

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click the **2D** button.
- 3 Click **Next**.
- 4 In the **Add physics** tree, select **Mathematics>PDE Interfaces>General Form PDE (g)**.
- 5 Click **Add Selected**.
- 6 In the **Number of dependent variables** edit field, type 2.
- 7 Click **Next**.
- 8 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.
- 9 Click **Finish**.

GEOMETRY 1

Circle 1

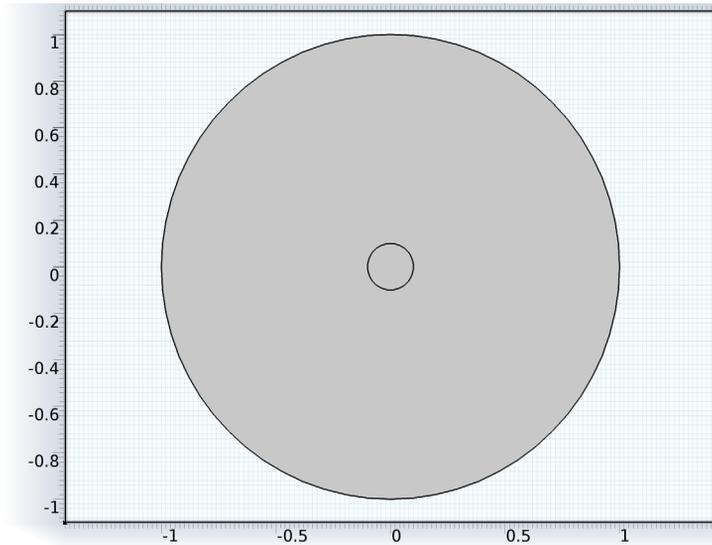
- 1 In the **Model Builder** window, right-click **Model 1>Geometry 1** and choose **Circle**.
- 2 Click the **Build Selected** button.

Circle 2

- 1 Right-click **Geometry 1** and choose **Circle**.
- 2 Go to the **Settings** window for Circle.
- 3 Locate the **Size and Shape** section. In the **Radius** edit field, type 0.1.
- 4 Click the **Build Selected** button.

Form Union

- 1 In the **Model Builder** window, right-click **Geometry 1**>**Form Union** and choose **Build Selected**.

**GLOBAL DEFINITIONS***Parameters*

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 Go to the **Settings** window for Parameters.
- 3 Locate the **Parameters** section. In the **Parameters** table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
alpha	1.449621256	Parameter for resistivity model
Jc	1.7e7[A/m ²]	Critical current density
I0	1e6[A]	Applied current
rho_air	1e6[ohm*m]	Resistivity of air
tau	0.02[s]	Time constant for applied current
Tc	92[K]	Critical temperature
dT	4[K]	Parameter for resistivity model
dJ	Jc/1e4	Parameter for resistivity model
E0	0.0836168[V/m]	Parameter for resistivity model

Next, add the variables used in the model, starting with the azimuthal unit vector.

Variables 1

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Variables**.
- 2 Go to the **Settings** window for Variables.
- 3 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION
ephix	$-y / (\text{sqrt}(x^2+y^2))$
ephiy	$x / (\text{sqrt}(x^2+y^2))$

DEFINITIONS

Variables 2

- 1 In the **Model Builder** window, right-click **Model 1 > Definitions** and choose **Variables**.
- 2 Go to the **Settings** window for Variables.
- 3 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
Jz_sc	$d(H_y, x) - d(H_x, y)$	Current density
normH_sc	$\text{sqrt}(H_x^2+H_y^2)$	Norm of the H-field
I1	$I_0 * (1 - \exp(-t/\tau))$	Applied current
Q_sc	$Ez_sc * Jz_sc$	Generated heat in superconductor

Because the electric field is defined differently in the wire and in the air according to [Equation 3](#), you need two additional **Variables** nodes.

Variables 3

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
- 2 Go to the **Settings** window for Variables.
- 3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 1 only.
- 5 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION
Ez_sc	$\rho_{\text{air}} * Jz_sc$

Variables 4

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
- 2 Go to the **Settings** window for Variables.
- 3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 2 only.
- 5 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION
Ez_sc	$E0 * ((Jz_sc - Jc) * f1c2hs(Jz_sc - Jc - dJ, dJ) / Jc) ^{\alpha}$

Finally, add the value of the magnetic field to be used as the boundary condition on the external boundaries.

Variables 5

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
- 2 Go to the **Settings** window for Variables.
- 3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 1, 2, 5, and 8 only.
- 5 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION
H0phi	$I1 / (2 * \pi * \sqrt{x^2 + y^2})$

PDE

- 1 In the **Model Builder** window, click **Model 1 > PDE**.
- 2 Go to the **Settings** window for PDE.
- 3 Locate the **Units** section. From the **Dependent variable quantity** list, choose **Magnetic field (A/m)**.
- 4 In the **Unit** edit field, type $V * m^{-2}$.
- 5 Click to expand the **Dependent Variables** section.
- 6 In the **Field name** edit field, type H.

7 In the **Dependent variables** table, enter the following settings:

Hx
Hy

To edit the shape function used for the dependent variable, the **Discretization** section must be visible.

8 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

9 Go to the **Settings** window for PDE.

10 Click to expand the **Discretization** section.

11 From the **Shape function type** list, choose **Curl**.

General Form PDE I

1 In the **Model Builder** window, expand the **PDE** node, then click **Model I>General Form PDE I**.

2 Go to the **Settings** window for General Form PDE.

3 Locate the **Conservative Flux** section. Specify the first Γ vector as

0	x
Ez_sc	y

4 Specify the 2nd Γ vector as

-Ez_sc	x
0	y

5 Locate the **Source Term** section. In the f edit-field array, type 0 on the first row.

6 In the f edit-field array, type 0 on the 2nd row.

7 Locate the **Damping or Mass Coefficient** section. In the d_a edit-field array, type μ_0_const in the first column of the first row.

8 In the d_a edit-field array, type μ_0_const in the 2nd column of the 2nd row.

Here, μ_0const is a predefined COMSOL Multiphysics constant for the permeability of vacuum.

Apply a Dirichlet boundary condition to the magnetic field on the exterior boundaries.

Dirichlet Boundary Condition 1

- 1 In the **Model Builder** window, right-click **PDE** and choose **Dirichlet Boundary Condition**.
- 2 Select Boundaries 1, 2, 5, and 8 only.
- 3 Go to the **Settings** window for Dirichlet Boundary Condition.
- 4 Locate the **Dirichlet Boundary Condition** section. In the r_1 edit field, type $H0phi*ephix$.
- 5 In the r_2 edit field, type $H0phi*ephiy$.

MESH 1

Free Triangular 1

In the **Model Builder** window, right-click **Model 1>Mesh 1** and choose **Free Triangular**.

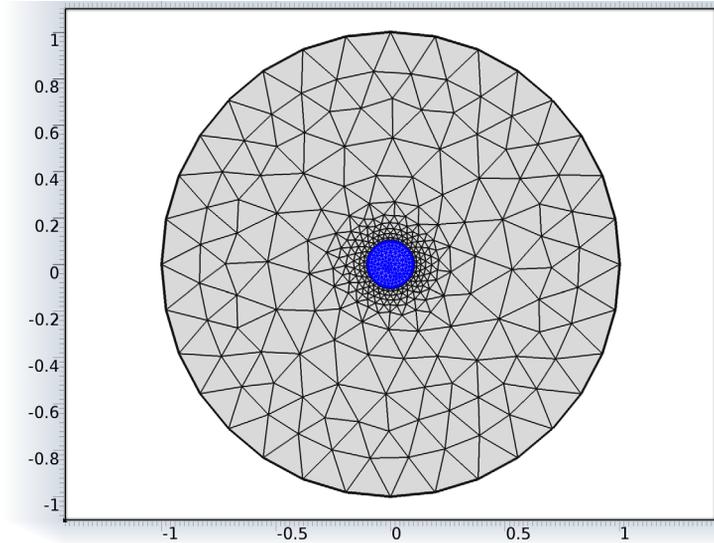
Size 1

- 1 In the **Model Builder** window, right-click **Mesh 1>Free Triangular 1** and choose **Size**.
- 2 Go to the **Settings** window for Size.
- 3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 2 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 7 In the associated edit field, type 0.02.

Size

- 1 In the **Model Builder** window, click **Model 1>Mesh 1>Size**.
- 2 Go to the **Settings** window for Size.
- 3 Locate the **Element Size** section. From the **Predefined** list, choose **Coarse**.

4 Click the **Build All** button.



STUDY 1

Step 1: Time Dependent

- 1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.
- 2 Go to the **Settings** window for Time Dependent.
- 3 Locate the **Study Settings** section. Click the **Range** button.
- 4 Go to the **Range** dialog box.
- 5 In the **Stop** edit field, type 0.1.
- 6 In the **Step** edit field, type 0.005.
- 7 Click the **Replace** button.
- 8 Go to the **Settings** window for Time Dependent.
- 9 Locate the **Study Settings** section. Select the **Relative tolerance** check box.
- 10 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.

Solver 1

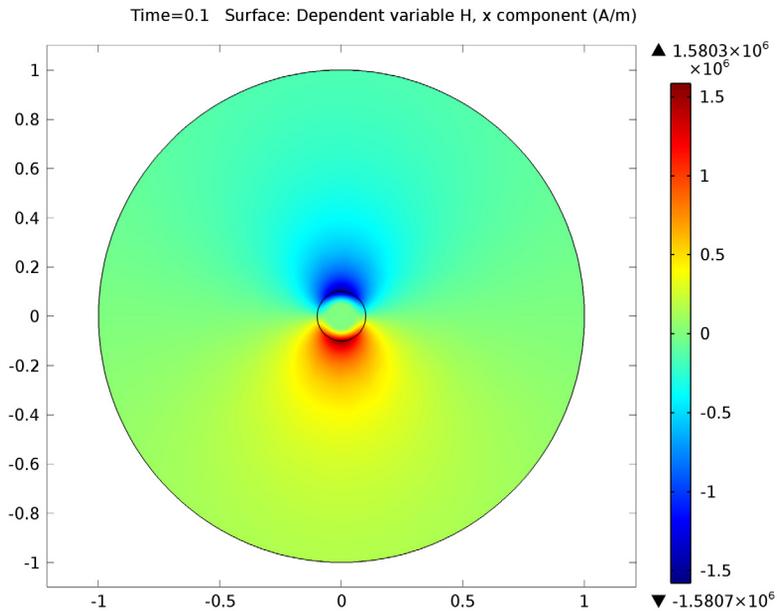
- 1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1** node, then click **Time-Dependent Solver 1**.
- 2 Go to the **Settings** window for Time-Dependent Solver.

- 3 Click to expand the **Time Stepping** section.
- 4 Select the **Initial step** check box.
- 5 In the associated edit field, type $1e-9$.
- 6 Select the **Maximum step** check box.
- 7 In the associated edit field, type $1e-3$.
- 8 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

2D Plot Group 1

When the computation is finished, the default plot shows the x -component of the magnetic field.

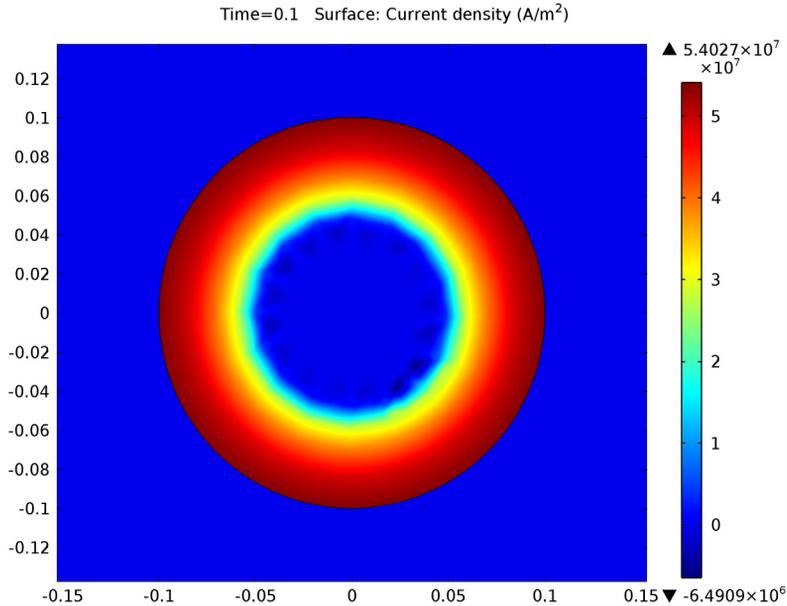


The following instructions explain how to produce the current density plot.

2D Plot Group 2

- 1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.
- 2 Right-click **Results>2D Plot Group 2** and choose **Surface**.
- 3 Go to the **Settings** window for Surface.
- 4 In the upper-right corner of the **Expression** section, click **Replace Expression**.

- 5 From the menu, choose **Definitions>Current density (Jz_sc)**.
- 6 Click the **Plot** button.
- 7 Click the **Zoom In** button on the Graphics toolbar two or three times to get a closer view of the wire.



Under the **Export** node, it is possible to create an animation of the evolution of the current density distribution.

Export

- 1 In the **Model Builder** window, right-click **Results>Export** and choose **Player**.
- 2 Go to the **Settings** window for Player.
- 3 Locate the **Scene** section. From the **Subject** list, choose **2D Plot Group 2**.
- 4 Click the **Generate Frame** button.
- 5 Right-click **Export>Player 1** and choose **Play**.