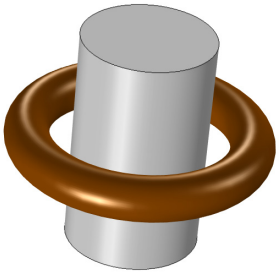
Eddy Currents in a Cylinder

Introduction

To help you understand how to create models using the AC/DC Module, this section walks through an example in great detail. You can apply these techniques to all the models in this module, other optional modules, or even the many models that ship with the base COMSOL Multiphysics package.

The example model concerns an AC coil surrounding a metal cylinder (core), and the coil induces eddy currents in the cylinder. It illustrates how to use the Coil Domain features to model the coil in different ways.



Model Definition

Due to the cylindrical symmetry of the problem, a 2D axisymmetric geometry is used. The modeling plane is the *rz*-plane; the horizontal axis represents the *r*-axis, and the vertical axis represents the *z*-axis. In this plane, the core appears as a rectangle and the coil as a circle. To obtain the actual 3D geometry, revolve the 2D axisymmetric geometry about the *z*-axis.

The physics used in this model is Magnetic Fields, with a Frequency Domain study type. The fundamental equations involved are explained in the following sections.

DOMAIN EQUATIONS

The dependent variable in this physics is the azimuthal component of magnetic vector potential, **A**, which, in frequency domain, obeys the relation:

©2013 COMSOL

$$(j\omega\sigma - \omega^2\varepsilon)A_\varphi + \nabla \times (\mu^{-1}\nabla \times A_\varphi) = J_\varphi^e \quad (1)$$

where ω is the angular frequency, σ is the electric conductivity, μ is the permeability, ε is the permittivity, and J_φ^e denotes the current density due to an external source. One way to define the current source is to specify a distributed current density in the right-hand side of the above equation. This current density gives rise to a current I as defined by:

$$\int_S \mathbf{J}^e \cdot d\mathbf{s} = I \quad (2)$$

BOUNDARY CONDITIONS

This model requires boundary conditions for the exterior boundary and the symmetry axis, and to specify boundary currents when applicable. The physics automatically applies a boundary condition corresponding to zero magnetic flux through the exterior boundary, by setting the vector potential to zero. On the symmetry axis, a suitable symmetry boundary condition is applied.

COIL GROUP DOMAINS

The Magnetic Fields interface provides features, called Coil Group Domain, that can conveniently be used to model coils in 2D and 2D axisymmetric geometries.

The Single-Turn Coil Domain models a single winding of a homogeneous metallic conductor, in which the current density can be non-uniform, due to the axial symmetry or skin effect. Note that applying the Single-Turn Coil Domain feature to multiple domains effectively consists in having multiple coils in parallel.

The Multi-Turn Coil Domain is used to model a great number of tiny wires wound together. In this approximation, the current density is uniform, and no conduction currents appear in the domain. From this follows that the material used to model this domain should reflect the properties of the insulator covering the wires, rather than of the metal constituting the windings.

A third Coil Domain feature is provided, that is not used in this model. The Coil Group Domain feature is identical to the Single-Turn Coil Domain, with the important difference that applying this feature to multiple domains connects the resulting coils in series.

 ©2013 COMSOL

Results and Discussion

The eddy currents induced in the cylinder are shown in the following figures. The induced currents on the cylinder are of comparable magnitude, due to the choice of the parameter, but the current density in the coil is different. Figure 1 presents the results obtained with a Single-Turn Coil Domain feature. The current density is not uniform and is greater in the inner part of the coil (the region closer to the z axis) and close to the surface (due to the skin effect).

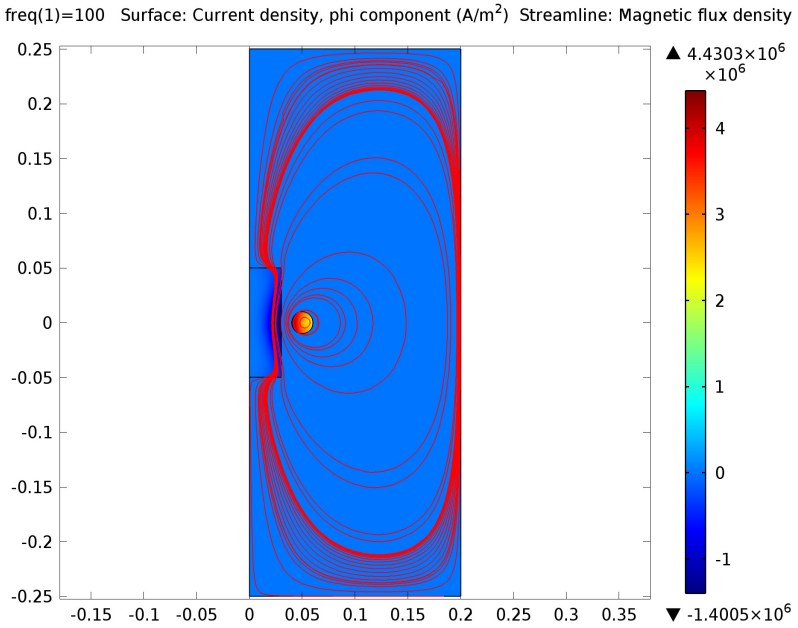


Figure 1: Results using a Single-Turn Coil Domain feature.

In Figure 2, the Multi-Turn Coil Domain feature is used. While the effect on the cylinder is similar to the one obtained with a Single Turn Coil Domain, the current density in the coil is uniform, according to the coil approximation stated above.

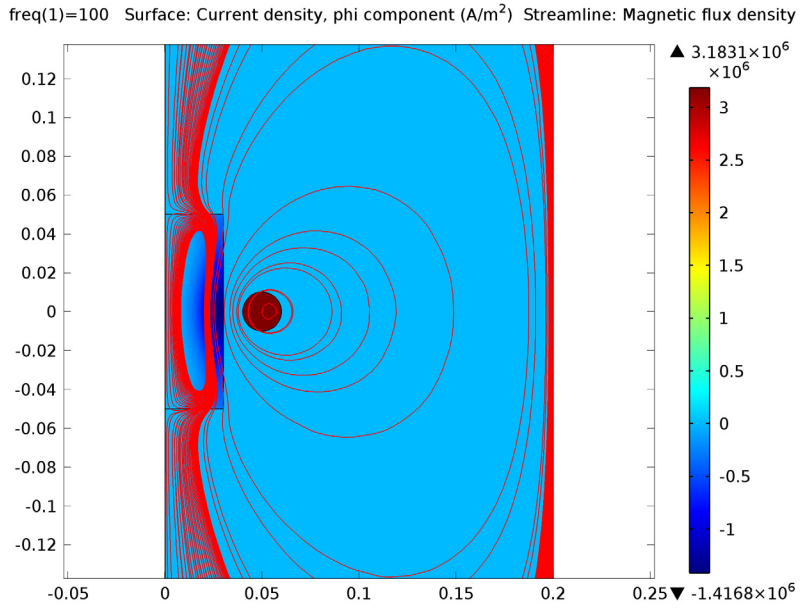


Figure 2: Results using a Multi-Turn Coil Domain feature.

Model Library path: `ACDC_Module/Inductive_Devices_and_Coils/coil_eddy_currents`

The Model Library path shows the location of the Model MPH-file. You can open it directly from the **Model Library** (accessible from the **View** menu) and browsing to **AC/DC Module>Inductive Devices and_Coils>coil eddy currents**.

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D axisymmetric** button.

©2013 COMSOL

3 Click **Next**.

4 In the **Add physics** tree, select **AC/DC>Magnetic Fields (mf)**.

5 Click **Add Selected**.

6 Click **Next**.

7 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.

8 Click **Finish**.

GEOMETRY 1

The following instructions explain how to build the model geometry.

Rectangle 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `0.2`.

4 In the **Height** edit field, type `0.5`.

5 Locate the **Position** section. In the **z** edit field, type `-0.25`.

6 Click the **Build Selected** button.

Rectangle 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `0.03`.

4 In the **Height** edit field, type `0.1`.

5 Locate the **Position** section. In the **z** edit field, type `-0.05`.

6 Click the **Build Selected** button.

Circle 1

1 Right-click **Geometry 1** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `0.01`.

4 Locate the **Position** section. In the **r** edit field, type `0.05`.

5 Click the **Build All** button.

This concludes the construction of the geometry. The next step is the definition of the material properties. Define the materials constituting the coil and the core.

6 Click the **Zoom Extents** button on the Graphics toolbar.

MATERIALS

Material 1

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.

2 Select Domain 3 only.

3 In the **Material** settings window, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Name	Value
Electrical conductivity	sigma	`3.7e7[S/m]`
Relative permittivity	epsilonr	1
Relative permeability	mur	1

5 Right-click **Model 1>Materials>Material 1** and choose **Rename**.

6 Go to the **Rename Material** dialog box and type `Coil` in the **New name** edit field.

7 Click **OK**.

Material 2

1 Right-click **Materials** and choose **Material**.

2 Select Domain 2 only.

3 In the **Material** settings window, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Name	Value
Electrical conductivity	sigma	`3.7e7[S/m]`
Relative permittivity	epsilonr	1
Relative permeability	mur	1

5 Right-click **Model 1>Materials>Material 2** and choose **Rename**.

6 Go to the **Rename Material** dialog box and type `Core` in the **New name** edit field.

7 Click **OK**.

The third material needed for the model is Air, used for the domain surrounding the core and the coil. The material parameters for air are already available in COMSOL Multiphysics, and can be accessed using the Material Browser.

 ©2013 COMSOL

Material Browser

1 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** settings window, In the tree, select **Built-In>Air**.

3 Click **Add Material to Model**.

Air

1 In the **Model Builder** window, under **Model 1>Materials** click **Air**.

2 Select Domain 1 only.

MAGNETIC FIELDS

The next step consists in setting up of the physics interface. The Magnetic Fields interface automatically provides default domain and boundary conditions. The following instructions explain how to apply a current density using a Single-Turn Coil Domain feature.

Single-Turn Coil 1

1 In the **Model Builder** window, right-click **Magnetic Fields** and choose **Single-Turn Coil**.

2 Select Domain 3 only.

3 In the **Single-Turn Coil** settings window, locate the **Single-Turn Coil** section.

4 In the I_{coil} edit field, type `1[kA]`.

No further actions are required in the physics interface. The next step is the mesh generation. To better resolve the induced current density in the core and the coil, use a fine mesh.

MESH 1

1 In the **Model Builder** window, under **Model 1** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Fine**.

4 Click the **Build All** button.

The last step consists in setting up the study. In this case you need to provide an operating frequency for the Frequency Domain study.

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.

2 In the **Frequency Domain** settings window, locate the **Study Settings** section.

3 In the **Frequencies** edit field, type `100[Hz]`.

4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

When the solution process is completed, a default plot showing the norm of the magnetic flux density is generated. You can add more plots to visualize other quantities. The following instructions illustrate how to plot the current density and the streamlines of the magnetic flux density.

RESULTS

2D Plot Group 3

1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.

2 Right-click **2D Plot Group 3** and choose **Surface**.

3 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Magnetic Fields>Currents and charge>Current density>Current density, phi component (mf.Jphi)**.

4 In the **Model Builder** window, right-click **2D Plot Group 3** and choose **Streamline**.

5 In the **Streamline** settings window, locate the **Streamline Positioning** section.

6 From the **Positioning** list, choose **Start point controlled**.

7 In the **Points** edit field, type `15`.

8 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Magnetic Fields>Magnetic>Magnetic flux density (mf.Br,mf.Bz)**.

9 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

10 Click the **Plot** button.

11 Click the **Zoom In** button on the Graphics toolbar.

The plot shows the azimuthal current density in the core and in the coil. The current density in the coil is greater in the inner region. Moreover, due to the skin effect, it is more concentrated at the surface.

The following instructions illustrate how to modify the model to use a Multi-Turn Coil Domain for the excitation. The first step is to change the material model to an insulator. In this case, use air.

MATERIALS

Air

1 In the **Model Builder** window, under **Model 1>Materials** click **Air**.

2 Select Domains 1 and 3 only.

 ©2013 COMSOL

Before adding a Multi-Turn Coil Domain feature you must disable the Single-Turn Coil Domain.

MAGNETIC FIELDS

Single-Turn Coil 1

In the **Model Builder** window, under **Model 1>Magnetic Fields** right-click **Single-Turn Coil 1** and choose **Disable**.

Multi-Turn Coil 1

1. Right-click **Magnetic Fields** and choose **Multi-Turn Coil**.
2. Select Domain 3 only.

   Specify the conductivity of the metal constituting the coil wires.
3. In the **Multi-Turn Coil** settings window, locate the **Multi-Turn Coil** section.
4. In the σ_{coil} edit field, type `3e7[S/m]`.

   To obtain comparable effects on the core, set the number of windings to 1000 and apply a current of 1 A.
5. In the N edit field, type `1000`.
6. In the I_{coil} edit field, type `1[A]`.

STUDY 1

1. In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

After the solution process, update the plot group to visualize the new results.

RESULTS

2D Plot Group 3

1. In the **Model Builder** window, under **Results** right-click **2D Plot Group 3** and choose **Plot**.
2. Click the **Zoom Extents** button on the Graphics toolbar.
3. Click the **Zoom In** button on the Graphics toolbar.

The induced current density in the core is comparable to the previous case; the current density in the coil, on the other hand, is uniform.

©2013 COMSOL