An Example—Eddy Currents

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 first example model concerns an AC coil surrounding a metal cylinder, and the coil induces eddy currents in the cylinder. It illustrates how to examine a system using several different approaches. You can model the coil with or without the skin effect, and it shows how varying the frequency of the current source also alters the depth of the skin effect.

Model Definition

To build this model, work with the axisymmetric Azimuthal Induction Currents, Vector Potential application mode (in the Quasi-Statics, Magnetic folder) in the time-harmonic formulation. The model represents the cylinder as a rectangle and the coil as a circle. The modeling plane is the *rz*-plane; the horizontal axis represents the *r*-axis, and the vertical axis represents the *z*-axis. To obtain the actual 3D geometry, revolve the 2D geometry about the *z*-axis.

DOMAIN EQUATIONS

The dependent variable in this application mode is the azimuthal component of magnetic vector potential, $\mathbf{A}$, which obeys the relation:

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

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:

$$\int_S \mathbf{J}^{\mathrm{e}} \cdot d\mathbf{s} = I$$

BOUNDARY CONDITIONS

This model requires boundary conditions for the exterior boundary and the symmetry axis, and to specify boundary currents when applicable. You can apply a condition corresponding to zero magnetic flux through the exterior boundary by setting the vector potential to zero. Next, give the symmetry boundary a symmetry condition. You can also specify the applied current source using equivalent surface currents:

$$\oint_C (\mathbf{n} \times \mathbf{J}_s) \cdot d\mathbf{l} = I$$

Model Library path: `ACDC_Module/Tutorial_Models/coil_eddy_currents`

The Model Library path shows the location of the Model MPH-file. You can open it directly from the **Model Navigator** by clicking the **Model Library** tab and browsing to **AC/DC Module>Tutorial Models>coil eddy currents**.

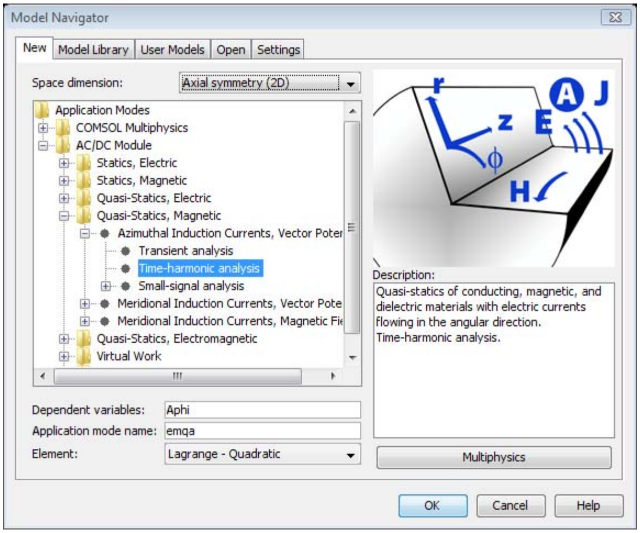
Coil Without Skin Effect

Begin this study of induced currents by modeling a current-carrying coil without skin effect.

MODEL NAVIGATOR

1. Begin a new COMSOL Multiphysics session by invoking the **Model Navigator**.
2. On the **New** page, select **Axial symmetry 2D** from the **Space dimension** list.
3. In the list of application modes, click on **AC/DC Module**, then select **Quasi-Statics, Magnetic>Azimuthal Induction Currents, Vector Potential** and finally **Time-harmonic analysis**.

4 Click **OK** to close the **Model Navigator**.

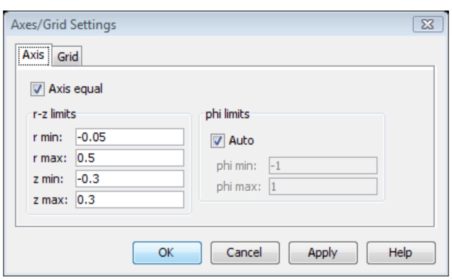


OPTIONS AND SETTINGS

Start the modeling session by adjusting the drawing area to hold the geometry you plan to draw. Another aid in making the simulation easier is to define variables for later use when defining the problem.

1 Select **Axes/Grid Settings** from the **Options** menu to open the **Axes/Grid Settings** dialog box.

2 On the **Axis** page, type `-0.05` and `0.5` in the **r min** and **r max** edit fields. Then set the *z*-axis limits to `-0.3` and `0.3`.



3 To manually define new grid settings, first click the **Grid** tab and then clear the **Auto** check box.

4 Type `0.05` in the **r spacing** edit field and type the value `0.03` in the **Extra r** edit field. Set the value in the **z spacing** edit field to `0.05` and add two extra grid lines by typing `-0.01 0.01` in the **Extra z** edit field.

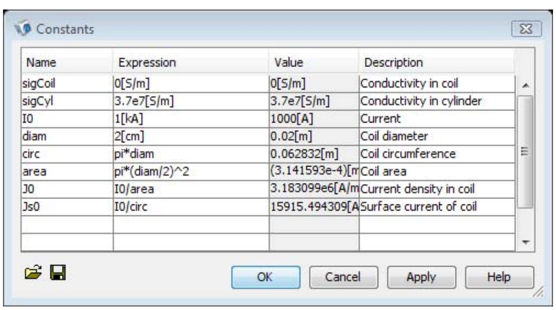
5 Click **Apply** to see the effects of the new settings. Notice that the interface adjusts the *r*-axis settings to maintain the correct aspect ratio. Click **OK** to close the dialog box.

6 To define global constants for the model, select **Constants** from the **Options** menu. Doing so opens the **Constants** dialog box.

7 Enter values in the **Name**, **Expression**, and (optionally) **Description** edit fields according to the following table:

NAME	EXPRESSION	DESCRIPTION
`sigCoil`	`0[S/m]`	`Conductivity in coil`
`sigCyl`	`3.7e7[S/m]`	`Conductivity in cylinder`
`I0`	`1[kA]`	`Current`
`diam`	`2[cm]`	`Coil diameter`
`circ`	`pi*diam`	`Coil circumference`
`area`	`pi*(diam/2)^2`	`Coil area`
`J0`	`I0/area`	`Current density in coil`
`Js0`	`I0/circ`	`Surface current of coil`

The numerical values used in the model are now visible in the dialog box.



8 Click **OK**.

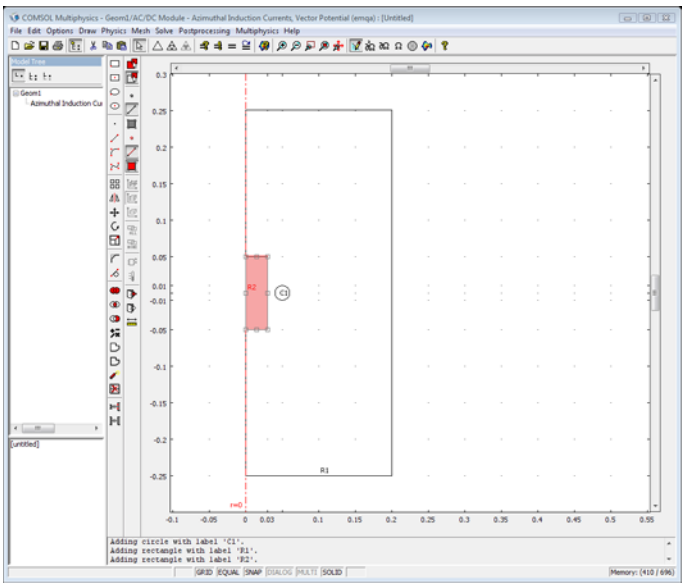
GEOMETRY MODELING

Now define the structure's geometry using the CAD tools built into COMSOL Multiphysics.

1 Start by drawing a circle that represents the coil. To do so, go to the Draw menu, point to **Draw Objects**, and then choose **Ellipse/Circle (Centered)**; alternately, go to the Draw toolbar on the left of the main drawing area and click the **Ellipse/Circle (Centered)** button. Click the right mouse button at (0.05, 0) and move the cursor to (0.05, 0.01) and then release the button. This action creates the desired circle.

2 Click the **Rectangle/Square** button on the Draw toolbar or choose the corresponding entry on the **Draw** menu (**Draw Objects>Rectangle/Square**), then using the left mouse button draw a rectangle with opposite corners at (0, –0.25) and (0.2, 0.25).

3 Choose **Draw>Draw Objects>Rectangle/Square** once again and create a new rectangle from (0, –0.05) to (0.03, 0.05). The use of extra grid lines together with the snap functionality makes this task easier to accomplish.

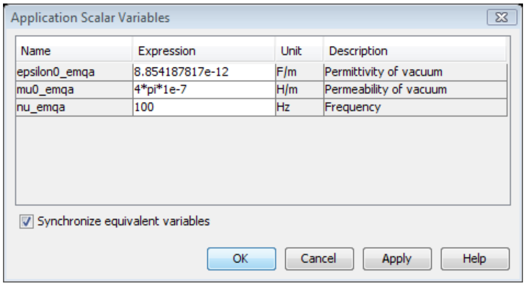


4 Double-click **GRID** in the status bar at the bottom of the window to hide the grid lines.

PHYSICS SETTINGS

Scalar Variables

1 The predefined variables specific to the active application mode are called *application scalar variables*. Just as you can do with global variables, you can use these in any expression for physical quantities, boundary conditions, or postprocessing entities. Open the corresponding dialog box by selecting **Scalar Variables** from the **Physics** menu.

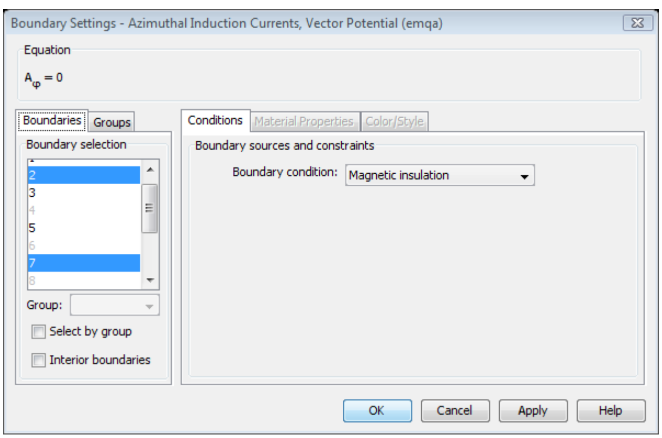


2 The current has a frequency of 100 Hz; enter the value `100` in the corresponding edit field in the **Expression** column on the **nu_emqa** row. This model uses the default values of the permittivity and permeability of vacuum, so leave these fields untouched.

3 Click **OK** to close the dialog box.

Boundary Conditions

1 Open the **Boundary Settings** dialog box by selecting **Boundary Settings** from the **Physics** menu.



2 Enter the boundary conditions according to the following table:

SETTING	BOUNDARIES 1, 3, 5	BOUNDARIES 2, 7, 9
Boundary condition	Axial symmetry	Magnetic insulation

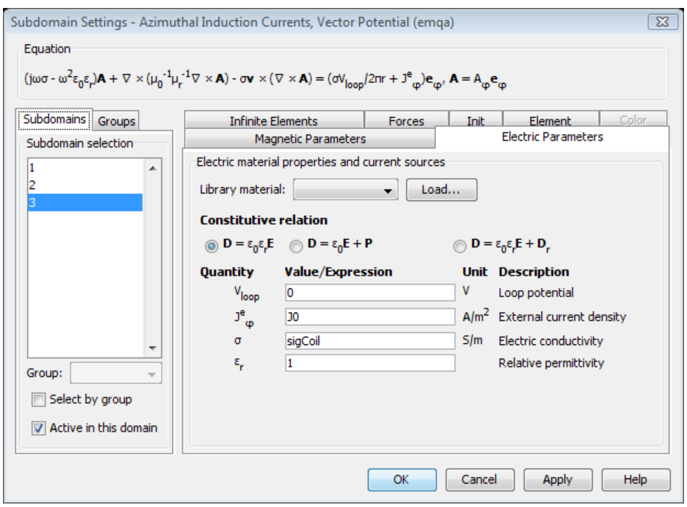
Note: You can select a boundary either in the **Boundary selection** list or by clicking on it in the main drawing area. To select several boundaries simultaneously, use the Shift and Ctrl keys.

3 Click **OK**.

Boundaries 1, 3, and 5 make up the vertical boundary along the *z*-axis, and the axial symmetry boundary condition makes certain the solution is symmetric around this axis. The boundary condition at the other three boundaries (2, 7, and 9) sets the magnetic potential, A_φ, to zero along that boundary.

Subdomain Settings

1 From the **Physics** menu, choose **Subdomain Settings**. Click the **Electric Parameters** tab.



2 The domain properties for this model appear in the following table. Use default values for any properties not supplied.

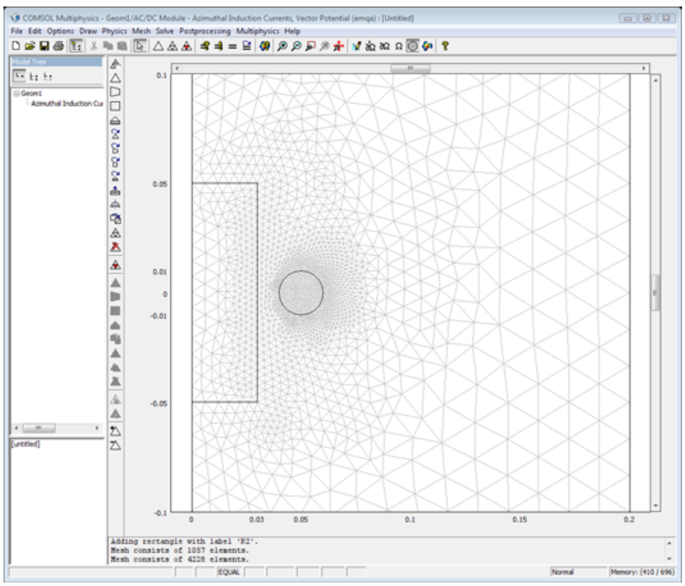
SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2	SUBDOMAIN 3
σ	0	sigCyl	sigCoil
J^e_φ	0	0	J0

3 Click **OK**.

MESH GENERATION

In this model, as in many others dealing with electromagnetic phenomena, the effects on fields near the interfaces between materials are of special interest. To get accurate results make sure to generate a very fine mesh in these areas. To do so in this case, refine the mesh one time.

1. To generate a mesh, choose **Initialize Mesh** from the **Mesh** menu, or use the corresponding button on the Main toolbar.
2. Choose **Refine Mesh** from the **Mesh** menu or use the corresponding button on the Main toolbar.
3. To better see the mesh in the region of interest, choose **Zoom>Zoom Window** from the **Options** menu. You can now draw a rectangular window around the coil and the cylinder to get a better view.



COMPUTING THE SOLUTION

Select **Solve Problem** from the **Solve** menu.

POSTPROCESSING AND VISUALIZATION

After solving the problem, the software automatically displays a surface plot for the dependent variable, in this case, the magnetic vector potential. The buttons on the Plot toolbar allow you to generate other types of plots.

To change the default plot parameters follow this procedure:

1 Open the **Plot Parameters** dialog box by choosing **Plot Parameters** from the **Postprocessing** menu.

2 On the **General** page, select the **Surface** and **Streamline** check boxes in the **Plot type** area.

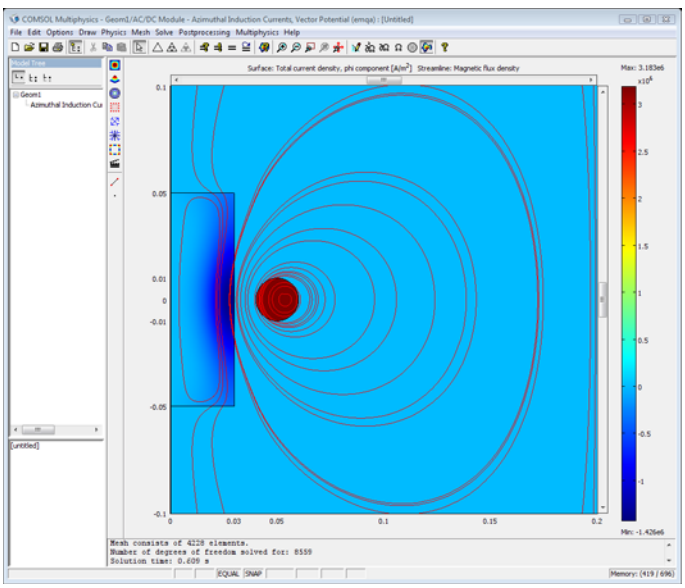
3 Click the **Surface** tab.

4 From the **Predefined quantities** list on the **Surface Data** page, select **Azimuthal Induction Currents, Vector Potential (emqa)>Total current density, phi component**.

5 Click the **Streamline** tab.

6 From the **Predefined quantities** list on the **Surface Data** page, select **Magnetic flux density**.

7 Click **OK** to generate the plot.



The plot shows the eddy currents induced in the cylinder and the constant current density in the coil.

Coil With Skin Effect

If the current-carrying coil is homogeneous, you know that a skin effect prevails, and it is easy to model this effect. In the previous example where the conductivity was zero in the coil, you prescribed the current density. Now all you must do is set a conductivity in the current-carrying coil.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Constants**.

2 Select `sigCoil` in the variable list and then type `3.7e7` in the corresponding **Expression** edit field.

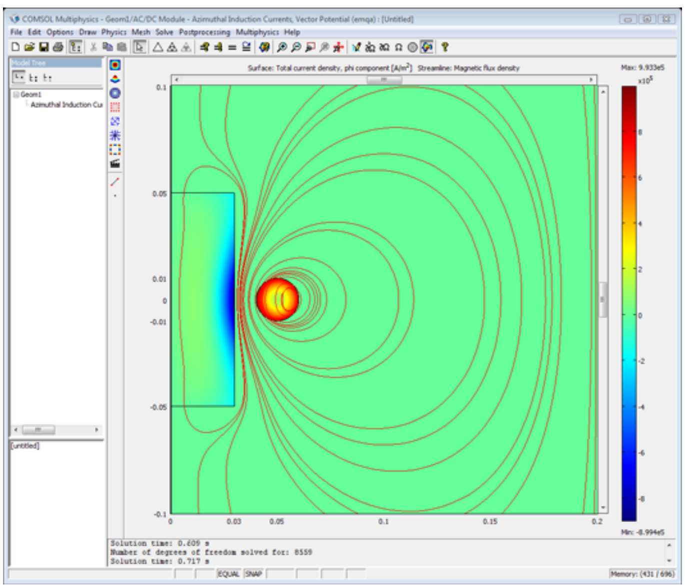
3 Click **OK**.

MESH GENERATION

The existing mesh is adequate, so there is no need to generate a new one.

COMPUTING THE SOLUTION

From the **Solve** menu, select **Solve Problem**; alternatively click the **Solve** button on the Main toolbar.



The skin effect in the coil is clearly visible.