

# Planar PCB Coil

## Introduction

Planar coils on printed-circuit board (PCB) are widely used for instance in radio-frequency identification, wireless charging and micro-motor devices. This tutorial model shows how to use the **Electric Currents in Shells** physics interface together with the **Magnetic Field** physics interface to compute the resistance and inductance of a planar PCB coil. The **Electric Currents in Shells** interface offers the advantage of drawing the entire coil layout as flat and on the same plane. A coil with three turns and two vias as represented in Figure 1 is modeled but the model showcases functionality that is applicable to a wide range of geometric configurations.



Figure 1: The structure of a simple PCB coil.

### Model Definition

The planar PCB coil is made of copper of 0.1 mm thickness. The current flows from one terminal on the top layer to the bottom layer through a via, back to the top layer through a second via and then to the other terminal that is grounded. It is worth noticing that the use of selections has made it possible to automatically identify all the crossings. The PCB substrate is typically made of epoxy resin, providing insulation between conducting layers. Since only the current conduction is of interest, the model uses air material settings (zero conductivity) also in the substrate.

Solving the electric currents, shell problem yields the electric potential drop along the conductor, and the lumped resistance R is computed according to Ohm's law

$$R = U/I \tag{1}$$

where U is the electric potential at the current terminal and I is the injected current (1 A in the current model setup). The computed surface current density flowing in the coil is then applied as the source term in the Ampère's law to compute the magnetic field distribution in the space surrounding the coil.

Note that here the electric currents have been computed for an open path. However, when dealing with magnetic fields, closing the current path is compulsory as current/charge conservation is inherent in Ampère's law. For this purpose, an extra magnetic insulation surface is added between the ground and current terminal. It will carry a reaction (numerically induced) surface current density, making the solution self-consistent with the divergence free nature of currents in Ampère's law.

The inductance can be derived as the ratio of linked magnetic flux to current. For the case of a single coil, the inductance can also be calculated using current and magnetic energy. In particular coil inductance can be expresses as

$$L = 2W_{\rm m}/I^2 \tag{2}$$

where  $W_{\rm m}$  is the total magnetic energy in the space.

## Results and Discussion

The model is solved using the default stationary solver. The plot for the electric potential distribution on the surface of the coil is shown in Figure 2. Note that this default plot automatically places the layers in their appropriate positions.



Figure 2: Electric potential distribution on the surface of the PCB coil.

The default magnetic field plot is shown in Figure 3. Arrows displaying the surface current density on the coil have been added.



Multislice: Magnetic flux density norm (T) Arrow Surface: Surface current density

Figure 3: The simulated surface current density of the coil and the magnetic field around it.

As a final step, the coil resistance and inductance are evaluated to 21.5 m $\Omega$  and 0.06  $\mu H$  respectively.

Application Library path: ACDC\_Module/Devices,\_Inductive/planar\_pcb\_coil

## 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 间 3D.

2 In the Select Physics tree, select AC/DC>Electric Fields and Currents> Electric Currents in Shells (ecis).

- 3 Click Add.
- 4 Click 🔿 Study.
- 5 In the Select Study tree, select General Studies>Stationary.
- 6 Click **M** Done.

#### 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>Copper.
- 4 Click Add to Global Materials in the window toolbar.
- 5 In the tree, select Built-in>FR4 (Circuit Board).
- 6 Right-click and choose Add to Global Materials.
- 7 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

#### **GLOBAL DEFINITIONS**

Upper Metalization

- I In the Model Builder window, under Global Definitions right-click Materials and choose Layered Material.
- 2 In the Settings window for Layered Material, type Upper Metalization in the Label text field.
- **3** Locate the **Layer Definition** section. In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Cu_up	Copper (mat1)	0.0	1E-4	2

Lower Metalization

- I Right-click Materials and choose Layered Material.
- 2 In the Settings window for Layered Material, type Lower Metalization in the Label text field.
- **3** Locate the **Layer Definition** section. In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Cu_down	Copper (mat I)	0.0	1E-4	2

Vias Metalization

I Right-click Materials and choose Layered Material.

- **2** In the **Settings** window for **Layered Material**, type Vias Metalization in the **Label** text field.
- 3 Locate the Layer Definition section. In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Cu_vias	Copper (mat I)	0.0	2E-4	2

#### GEOMETRY I

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.
- 3 From the Length unit list, choose mm.

Work Plane I (wp1)

In the **Geometry** toolbar, click 🖶 Work Plane.

Work Plane I (wpI)>Plane Geometry

In the Model Builder window, click Plane Geometry.

Work Plane I (wpl)>Polygon I (poll)

I In the Work Plane toolbar, click / Polygon.

- 2 In the Settings window for Polygon, locate the Coordinates section.
- **3** From the **Data source** list, choose **Vectors**.
- **4** In the **xw** text field, type -6 -6 -5.5 -5.5 -4 -4 4.5 4.5 -3 -3 3.5 3.5 -2 -2 -2 2.5 2.5 -1 -1 -0.5 -0.5 2 2 -1.5 -1.5 -1.5 3 3 -2.5 -2.5 4 4 -3.5 -3.5 -5.5 -5.5 -6.
- **5** In the **yw** text field, type 0.5 2 2 1.5 1.5 3.5 3.5 -4 -4 2.5 2.5 -3 -3 -2 1.5 1.5 -2 -2 -1 -1 -1.5 -1.5 1 1 -2 -2.5 -2.5 2 2 -3.5 -3.5 3 3 1 1 0.5 0.5.
- 6 Click 틤 Build Selected.
- **7** Click the |+| **Zoom Extents** button in the **Graphics** toolbar.

Work Plane I (wp1)>Circle I (c1)

- I In the Work Plane toolbar, click 🕑 Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type 0.5.
- 4 Locate the Position section. In the xw text field, type -0.75.
- 5 In the **yw** text field, type -0.75.
- 6 Click 틤 Build Selected.

Work Plane 1 (wp1)>Polygon 2 (pol2)

- I In the Work Plane toolbar, click / Polygon.
- 2 In the Settings window for Polygon, locate the Coordinates section.
- 3 From the Data source list, choose Vectors.
- **4** In the **xw** text field, type -6 -6 -5.5 -5.5 -4 -4 -5.5 -5.5.
- **5** In the **yw** text field, type -1.5 0 0 -0.5 -0.5 -1 -1 -1.5.

Work Plane I (wpI)>Circle 2 (c2)

- I In the Work Plane toolbar, click 😶 Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type 0.5.
- 4 Locate the **Position** section. In the **xw** text field, type -3.75.
- **5** In the **yw** text field, type -0.75.
- 6 In the Work Plane toolbar, click 📗 Build All.

Work Plane I (wp1)>Union I (uni1)

- I In the Work Plane toolbar, click 📕 Booleans and Partitions and choose Union.
- 2 Click in the Graphics window and then press Ctrl+A to select all objects.

Work Plane I (wp1)>Delete Entities I (del1)

I Right-click Plane Geometry and choose Delete Entities.



2 On the object unil, select Boundaries 15–17, 38, 39, and 42 only.

Work Plane 2 (wp2)

In the Model Builder window, right-click Geometry I and choose Work Plane.

Work Plane 2 (wp2)>Polygon 1 (pol1)

- I In the Work Plane toolbar, click / Polygon.
- 2 In the Settings window for Polygon, locate the Coordinates section.
- 3 From the Data source list, choose Vectors.
- **4** In the **xw** text field, type -3.5 -1 -1 -3.5.
- **5** In the **yw** text field, type -0.5 -0.5 -1 -1.
- 6 Click 틤 Build Selected.
- 7 Click the **Zoom Extents** button in the **Graphics** toolbar.

Work Plane 2 (wp2)>Circle 1 (c1)

- I In the Work Plane toolbar, click 🕑 Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type 0.5.
- 4 Locate the **Position** section. In the **xw** text field, type -0.75.
- 5 In the yw text field, type -0.75.

#### Work Plane 2 (wp2)>Circle 2 (c2)

- I In the Work Plane toolbar, click (•) Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type 0.5.
- 4 Locate the **Position** section. In the **xw** text field, type -3.75.
- 5 In the **yw** text field, type -0.75.
- 6 In the Work Plane toolbar, click 📳 Build All.

#### Work Plane 2 (wp2)>Union 1 (uni1)

- I In the Work Plane toolbar, click 💻 Booleans and Partitions and choose Union.
- 2 Click in the Graphics window and then press Ctrl+A to select all objects.

#### Work Plane 2 (wp2)>Delete Entities 1 (del1)

- I In the Model Builder window, right-click Plane Geometry and choose Delete Entities.
- **2** On the object **unil**, select Boundaries 1–3 and 6–8 only.





In the Model Builder window, right-click Geometry I and choose Work Plane.

Work Plane 3 (wp3)>Rectangle 1 (r1)

- I In the Work Plane toolbar, click 📃 Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.

- 3 In the Width text field, type 0.5.
- 4 In the **Height** text field, type 0.5.
- 5 Locate the Position section. In the xw text field, type -6.

#### Upper Layer

- I In the Model Builder window, under Component I (compl)>Geometry I click Work Plane I (wpl).
- 2 In the Settings window for Work Plane, type Upper Layer in the Label text field.
- **3** Locate the Selections of Resulting Entities section. Find the Cumulative selection subsection. Click New.
- 4 In the New Cumulative Selection dialog box, type Top in the Name text field.
- 5 Click OK.

#### Lower Layer

- I In the Model Builder window, under Component I (compl)>Geometry I click Work Plane 2 (wp2).
- 2 In the Settings window for Work Plane, type Lower Layer in the Label text field.
- **3** Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.
- 4 In the New Cumulative Selection dialog box, type Bottom in the Name text field.
- 5 Click OK.

#### Magnetic Insulation Layer

- In the Model Builder window, under Component I (compl)>Geometry I click
  Work Plane 3 (wp3).
- 2 In the Settings window for Work Plane, type Magnetic Insulation Layer in the Label text field.
- **3** Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.
- 4 In the New Cumulative Selection dialog box, type Insulation in the Name text field.
- 5 Click OK.

Now add a block that will constitute the domain in which to solve for the magnetic field around the coil.

Block I (blkI)

I In the **Geometry** toolbar, click 🗍 **Block**.

- 2 In the Settings window for Block, locate the Size and Shape section.
- 3 In the Width text field, type 18.
- 4 In the **Depth** text field, type 17.
- **5** In the **Height** text field, type **10**.
- 6 Locate the Position section. From the Base list, choose Center.
- 7 Click 틤 Build Selected.

Form Union (fin)

- I In the Geometry toolbar, click 📗 Build All.
- **2** Click the  $\leftarrow$  **Zoom Extents** button in the **Graphics** toolbar.

#### DEFINITIONS

Hide for Geometry I

- I In the Model Builder window, expand the Component I (compl)>Definitions node.
- 2 Right-click View I and choose Hide for Geometry.
- 3 In the Settings window for Hide for Geometry, locate the Selection section.
- 4 From the Geometric entity level list, choose Boundary.
- 5 On the object fin, select Boundaries 1, 2, and 4 only.



#### GEOMETRY I

Vias

- I In the Geometry toolbar, click 🝖 Selections and choose Explicit Selection.
- 2 In the Settings window for Explicit Selection, type Vias in the Label text field.
- **3** Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 On the object fin, select Boundaries 9 and 17 only.





- I In the Geometry toolbar, click 🔓 Selections and choose Union Selection.
- 2 In the Settings window for Union Selection, locate the Geometric Entity Level section.
- **3** From the **Level** list, choose **Boundary**.
- 4 In the Label text field, type PCB.
- 5 Locate the Input Entities section. Click + Add.
- 6 In the Add dialog box, in the Selections to add list, choose Top and Bottom.
- 7 Click OK.

Intersections and Vias

I In the Geometry toolbar, click 🔓 Selections and choose Intersection Selection.

- **2** In the Settings window for Intersection Selection, locate the Geometric Entity Level section.
- **3** From the Level list, choose **Boundary**.
- 4 In the Label text field, type Intersections and Vias.
- 5 Locate the Input Entities section. Click + Add.
- 6 In the Add dialog box, in the Selections to intersect list, choose Top and Bottom.
- 7 Click OK.

#### Intersections

- I In the Geometry toolbar, click 🔓 Selections and choose Difference Selection.
- 2 In the Settings window for Difference Selection, locate the Geometric Entity Level section.
- **3** From the Level list, choose **Boundary**.
- 4 In the Label text field, type Intersections.
- 5 Locate the Input Entities section. Click + Add.
- 6 In the Add dialog box, select Intersections and Vias in the Selections to add list.
- 7 Click OK.
- 8 In the Settings window for Difference Selection, locate the Input Entities section.
- 9 Click + Add.

10 In the Add dialog box, select Vias in the Selections to subtract list.

- II Click OK.
- 12 In the Geometry toolbar, click 🟢 Build All.

#### 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>Air.
- 4 Click the right end of the Add to Component split button in the window toolbar.
- 5 From the menu, choose Component I (compl).
- 6 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

#### MATERIALS

Air (mat3)

I In the Settings window for Material, locate the Geometric Entity Selection section.

- 2 From the Geometric entity level list, choose Domain.
- **3** From the Selection list, choose All domains.

#### Upper Metalization

- I In the Model Builder window, right-click Materials and choose Layers> Layered Material Stack.
- 2 In the Settings window for Layered Material Stack, type Upper Metalization in the Label text field.
- **3** Locate the **Boundary Selection** section. From the **Selection** list, choose **Top**.
- 4 Locate the Orientation and Position section. From the Position list, choose Bottom side on boundary.

Backing Under Upper Metalization

- I Right-click Upper Metalization and choose Layered Material.
- 2 In the Settings window for Layered Material, locate the Layer Definition section.
- **3** In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness (m)	Mesh elements
Backing	FR4 (Circuit Board) (mat2)	0.0	3E-4	2

- 4 In the Label text field, type Backing Under Upper Metalization.
- 5 Right-click Backing Under Upper Metalization and choose Move Up.

#### Upper Metalization

- I In the Model Builder window, under Component I (comp1)>Materials> Upper Metalization (stlmat1) click Layered Material Link I (stlmat1.stllmat1).
- 2 In the Settings window for Layered Material Link, type Upper Metalization in the Label text field.

Lower Metalization

- I In the Model Builder window, under Component I (compl) right-click Materials and choose Layers>Layered Material Stack.
- 2 In the Settings window for Layered Material Stack, type Lower Metalization in the Label text field.
- 3 Locate the Boundary Selection section. From the Selection list, choose Bottom.
- 4 Locate the Orientation and Position section. From the Position list, choose Bottom side on boundary.

#### Lower Metalization

- I In the Model Builder window, click Layered Material Link I (stlmat2.stllmat1).
- 2 In the Settings window for Layered Material Link, locate the Link Settings section.
- 3 From the Material list, choose Lower Metalization (Imat2).
- 4 In the Label text field, type Lower Metalization.

#### Intersections

- I In the Model Builder window, under Component I (compl) right-click Materials and choose Layers>Layered Material Stack.
- 2 In the Settings window for Layered Material Stack, type Intersections in the Label text field.
- **3** Locate the **Boundary Selection** section. From the **Selection** list, choose **Intersections**.
- 4 Locate the Orientation and Position section. From the Position list, choose Bottom side on boundary.

#### Lower Metalization

- I In the Model Builder window, click Layered Material Link I (stlmat3.stllmat1).
- 2 In the Settings window for Layered Material Link, locate the Link Settings section.
- 3 From the Material list, choose Lower Metalization (Imat2).
- 4 In the Label text field, type Lower Metalization.

#### Intersection Insulator

- I In the Model Builder window, right-click Intersections (stlmat3) and choose Layered Material.
- 2 In the Settings window for Layered Material, type Intersection Insulator in the Label text field.
- 3 Locate the Layer Definition section. In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness (m)	Mesh elements
Insulator	FR4 (Circuit Board) (mat2)	0.0	2E-4	2

#### Upper Metalization

- I Right-click Intersections (stImat3) and choose Layered Material Link.
- 2 In the Settings window for Layered Material Link, type Upper Metalization in the Label text field.

Vias

- I In the Model Builder window, under Component I (compl) right-click Materials and choose Layers>Layered Material Stack.
- 2 In the Settings window for Layered Material Stack, locate the Boundary Selection section.
- **3** From the **Selection** list, choose **Vias**.
- 4 In the Label text field, type Vias.
- **5** Locate the **Orientation and Position** section. From the **Position** list, choose **Bottom side on boundary**.

Lower Metalization

- I In the Model Builder window, under Component I (compl)>Materials>Vias (stlmat4) click Layered Material Link I (stlmat4.stllmat1).
- 2 In the Settings window for Layered Material Link, type Lower Metalization in the Label text field.
- 3 Locate the Link Settings section. From the Material list, choose Lower Metalization (Imat2).

Vias Metalization

- I In the Model Builder window, right-click Vias (stImat4) and choose Layered Material Link.
- 2 In the Settings window for Layered Material Link, locate the Link Settings section.
- 3 From the Material list, choose Vias Metalization (Imat3).
- 4 In the Label text field, type Vias Metalization.

#### Upper Metalization

- I Right-click Vias (stlmat4) and choose Layered Material Link.
- 2 In the Settings window for Layered Material Link, type Upper Metalization in the Label text field.

#### ELECTRIC CURRENTS IN SHELLS (ECIS)

- I In the Model Builder window, under Component I (compl) click Electric Currents in Shells (ecis).
- **2** In the **Settings** window for **Electric Currents in Shells**, locate the **Boundary Selection** section.
- **3** From the **Selection** list, choose **PCB**.
- 4 Locate the Shell Properties section. From the Shell type list, choose Layered shell.

#### Conductive Shell I

In the Model Builder window, expand the Component I (compl)>

**Electric Currents in Shells (ecis)>Conductive Shell I** node, then click **Conductive Shell I**.

#### Terminal I

- I In the Physics toolbar, click 🔙 Attributes and choose Terminal.
- 2 In the Settings window for Terminal, locate the Edge Selection section.
- 3 In the list, select 90 (not applicable).
- 4 From the Selection list, choose Manual.
- 5 Select Edge 12 only.
- **6** Locate the **Terminal** section. In the  $I_0$  text field, type 1.

#### Ground I

- I In the Physics toolbar, click 🔚 Edges and choose Ground.
- 2 In the Settings window for Ground, locate the Edge Selection section.
- **3** From the **Selection** list, choose **Manual**.
- 4 Select Edge 14 only.

#### Insulating Layer 1

- I In the Physics toolbar, click 📄 Boundaries and choose Insulating Layer.
- 2 In the Settings window for Insulating Layer, locate the Boundary Selection section.
- 3 From the Selection list, choose All boundaries.
- 4 Locate the Shell Properties section. Specify the Selection vector as

√ Backing - Backing Under Upper Metalization

Cu\_up - Upper Metalization

Insulating Layer 2

- I In the Physics toolbar, click 📄 Boundaries and choose Insulating Layer.
- 2 In the Settings window for Insulating Layer, locate the Boundary Selection section.
- 3 From the Selection list, choose All boundaries.
- 4 Locate the Shell Properties section. From the Layer list, choose Intersections (stImat3).
- **5** Specify the **Selection** vector as

Cu\_down - Lower Metalization

## √ Insulator - Intersection Insulator Cu\_up - Upper Metalization

Continuity I

- I In the Physics toolbar, click 🔚 Edges and choose Continuity.
- 2 In the Settings window for Continuity, locate the Layer Selection section.
- 3 From the Source list, choose Upper Metalization (stlmatl).
- 4 From the Destination list, choose Intersections (stImat3).

#### Continuity 2

- I In the Physics toolbar, click 🔚 Edges and choose Continuity.
- 2 In the Settings window for Continuity, locate the Layer Selection section.
- 3 From the Source list, choose Upper Metalization (stlmatl).
- 4 From the **Destination** list, choose **Vias (stlmat4)**.

#### Continuity 3

- I In the Physics toolbar, click 📄 Edges and choose Continuity.
- 2 In the Settings window for Continuity, locate the Layer Selection section.
- 3 From the Source list, choose Lower Metalization (stlmat2).
- 4 From the **Destination** list, choose **Intersections (stImat3)**.

#### Continuity 4

- I In the Physics toolbar, click 📄 Edges and choose Continuity.
- 2 In the Settings window for Continuity, locate the Layer Selection section.
- 3 From the Source list, choose Lower Metalization (stlmat2).
- 4 From the **Destination** list, choose **Vias (stlmat4)**.

#### ADD PHYSICS

- I In the Physics toolbar, click 🙀 Add Physics to open the Add Physics window.
- 2 Go to the Add Physics window.
- 3 In the tree, select AC/DC>Electromagnetic Fields>Magnetic Fields (mf).
- 4 Click Add to Component I in the window toolbar.
- 5 In the Physics toolbar, click 🙀 Add Physics to close the Add Physics window.

#### MAGNETIC FIELDS (MF)

Surface Current Density 1

- I Right-click Component I (comp1)>Magnetic Fields (mf) and choose Surface Current Density.
- 2 In the Settings window for Surface Current Density, locate the Surface Current Density section.
- **3** Specify the **J**<sub>s0</sub> vector as

ecis.JsX	x
ecis.JsY	у
ecis.JsZ	z

4 Locate the Boundary Selection section. From the Selection list, choose PCB.

#### Magnetic Insulation 2

I In the Physics toolbar, click 🔚 Boundaries and choose Magnetic Insulation.

2 In the Settings window for Magnetic Insulation, locate the Boundary Selection section.

**3** From the **Selection** list, choose **Insulation**.

#### MESH I

Free Tetrahedral I

In the Mesh toolbar, click \land Free Tetrahedral.

#### Size I

- I Right-click Free Tetrahedral I and choose Size.
- 2 In the Settings window for Size, locate the Element Size section.
- **3** Click the **Custom** button.
- **4** Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- 5 From the Selection list, choose PCB.
- 6 Locate the Element Size Parameters section.
- 7 Select the Maximum element size check box. In the associated text field, type 0.35.
- 8 Click 📗 Build All.

#### STUDY I

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

#### RESULTS

#### Electric Potential (ecis)

Electric potential distribution on the surface of the PCB coil.

#### Multislice I

- I In the Model Builder window, expand the Results>Magnetic Flux Density Norm (mf) node, then click Multislice I.
- 2 In the Settings window for Multislice, locate the Multiplane Data section.
- 3 Find the x-planes subsection. From the Entry method list, choose Number of planes.
- 4 In the Planes text field, type 0.
- 5 Find the y-planes subsection. From the Entry method list, choose Number of planes.
- 6 In the Planes text field, type 0.
- 7 In the Magnetic Flux Density Norm (mf) toolbar, click 💿 Plot.

#### Streamline Multislice I

In the Model Builder window, under Results>Magnetic Flux Density Norm (mf) right-click Streamline Multislice I and choose Delete.

Arrow Surface 1

- I In the Model Builder window, right-click Magnetic Flux Density Norm (mf) and choose Arrow Surface.
- 2 In the Settings window for Arrow Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)> Magnetic Fields>mf.scul.js0x,...,mf.scul.js0z Surface current density.
- 3 Locate the Coloring and Style section.
- 4 Select the Scale factor check box. In the associated text field, type 2e-4.
- 5 Locate the Arrow Positioning section. In the Number of arrows text field, type 1e4.
- 6 Locate the Coloring and Style section. From the Color list, choose Black.
- 7 In the Magnetic Flux Density Norm (mf) toolbar, click 💽 Plot.

The simulated surface current density of the coil and the magnetic field around it.

#### Global Evaluation 1

- I In the **Results** toolbar, click (8.5) **Global Evaluation**.
- 2 In the Settings window for Global Evaluation, click Add Expression in the upper-right corner of the Expressions section. From the menu, choose Component I (comp1)> Electric Currents in Shells>Terminals>ecis.V0\_I Terminal voltage V.

**3** Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
ecis.V0_1/ecis.I0_1	mΩ	Resistance
2*mf.intWm/ecis.I0_1^2	uH	Inductance

4 Click **=** Evaluate.