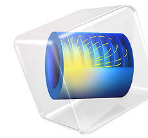


Resonant Spiral Coil in 3D

Introduction

This application presents a self-resonating, 5-turn spiral coil of copper wire with 1 mm square cross section. The wire is equipped with a thin epoxy varnish insulation layer, tightly wound and mounted on a printed circuit board (PCB). It is fed by lines on the reverse side of the PCB. The capacitive coupling between the consecutive turns separated by the geometrically thin varnish layer is modeled using a special boundary condition in the **Magnetic and Electric Fields** interface, thereby avoiding costly volumetric meshing. In order to efficiently resolve the small skin depth in the copper a **Boundary Layer** mesh is used. The modeling of thin insulation layers and a small skin depth effectively becomes intractable in 3D if traditional volumetric meshing has to be employed for capturing such effects. As shown in the axisymmetric variant of this model, Resonant Spiral Coil in 2D Axisymmetry, it is a challenging task already in 2D.

The purpose of the application is to investigate the self-resonance of the coil, that is sweeping the frequency through the regime where the coil reactance transitions from being inductive at frequencies below resonance to being capacitive at frequencies above resonance.

Model Definition

The model geometry consists of the spiral-shaped copper inductor with feed lines, the printed circuit board (PCB), and the surrounding air. Figure 1 shows the inductor and PCB domains used in the model. The outer diameter of the coil is about 20 mm.

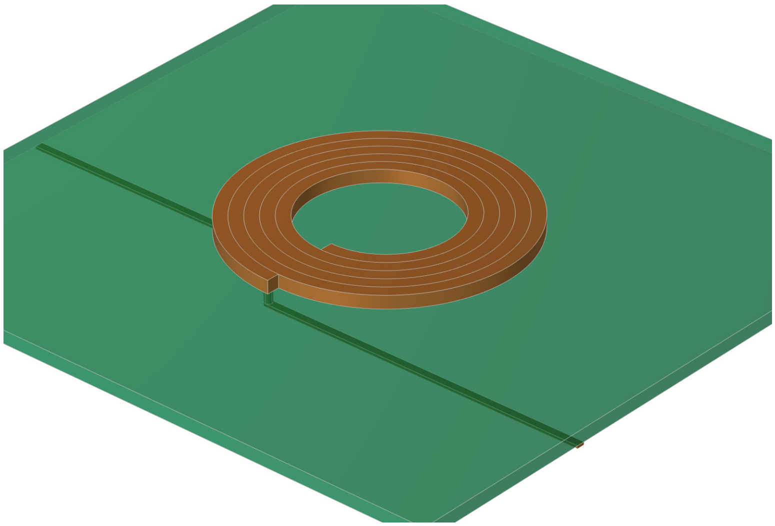


Figure 1: Spiral coil geometry including the PCB and feed lines on the back side.

The application uses the Magnetic and Electric Fields interface, taking electric and magnetically induced currents into account. This formulation, often referred to as an AV-formulation, solves both for the magnetic vector potential **A** and the electric potential *V*. The model is solved with a free gauge, using an iterative, residual minimizing method.

Using the definitions of the electric and magnetic potentials, the system of equations becomes

$$-\nabla \cdot ((j\omega\sigma - \omega^2\varepsilon_r\varepsilon_0)\mathbf{A} - (\sigma + j\omega\varepsilon_r\varepsilon_0)\nabla V - \mathbf{J}_e) = 0$$

$$(j\omega\sigma - \omega^2\varepsilon_r\varepsilon_0)\mathbf{A} + \nabla \times (\mu_\mathrm{r}^{-1}\mu_0^{-1}\nabla \times \mathbf{A}) + (\sigma + j\omega\varepsilon_r\varepsilon_0)\nabla V = \mathbf{J}_e$$

In the equations above, **A** denotes the magnetic vector potential, *V* the electric scalar potential, $\mathbf{J}_\mathrm{e}$ the externally generated or prescribed current density vector, σ the electrical conductivity, ε_0 the permittivity in vacuum, ε_r the relative permittivity, μ_0 the permeability in vacuum, and μ_r the relative permeability. The first equation is the equation of continuity, stating the conservation of charge. The second equation is Maxwell–Ampère's law. Because the model is excited using boundary constraints for the electric scalar potential, the externally generated current density, $\mathbf{J}_e$ is zero everywhere.

The application uses the **Terminal** boundary condition, which applies an electric potential of 5 mV at the end of one feed line and the **Ground** boundary condition at the end of the other feed line. A **Terminal** feature defines one entry in the admittance matrix, here a scalar since there is only one **Terminal**. The feed impedance Z is defined as the reciprocal value of the single Y_{11} component of the admittance matrix.

Using an AV-formulation allows for an interior boundary condition representing a thin, insulating or highly resistive, dielectric layer, relating the normal component of the total (displacement + conduction) current density to a discontinuity in the electric potential while the magnetic vector potential and the magnetic flux density are continuous over the boundary. This **Magnetic Continuity** boundary condition with a **Contact Impedance** subfeature is applied on all boundaries exterior to the copper domains save the end feeds where the **Terminal** and **Ground** conditions are applied.

Results

Figure 2 shows the current density in the coil at the resonance frequency. The skin and proximity effects are pronounced with the current flow concentrated to the edges of the turns.

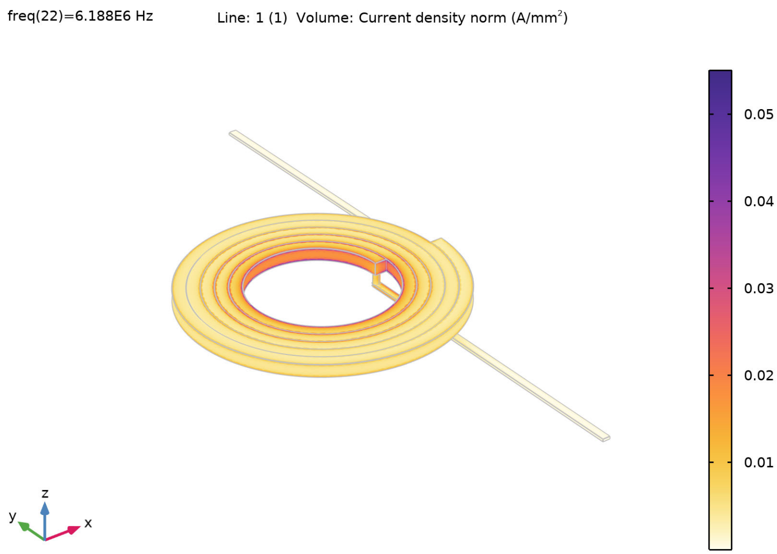


Figure 2: Current density in the coil exhibiting pronounced skin and proximity effects.

[Figure 3](#) displays the feed impedance of the coil where the reactance transitions from being inductive at frequencies below resonance to being capacitive at frequencies above resonance.

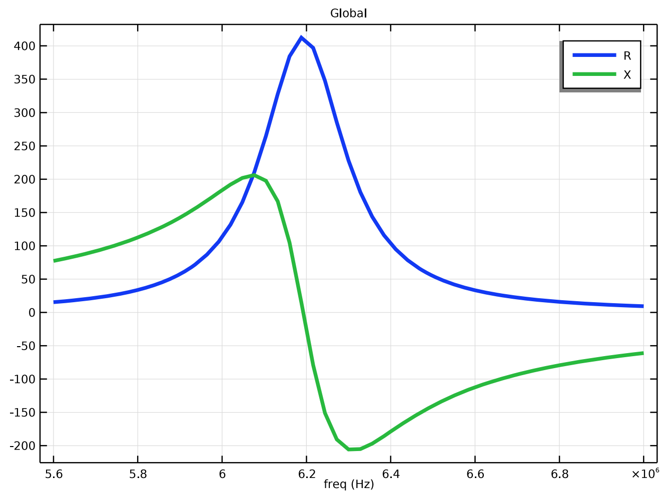


Figure 3: The feed impedance (resistance and reactance) is shown. Note the transition from inductive (positive) to capacitive (negative) reactance at the resonance frequency.

Application Library path: `ACDC_Module/Tutorials,_Coils/resonant_spiral_coil_3d`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **AC/DC>Electromagnetic Fields>Vector Formulations>Magnetic and Electric Fields (mef)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Frequency Domain**.

6 Click **Done**.

Define all the required parameters.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
`frq`	`6e6[Hz]`	6E6 Hz	
`delta`	`sqrt(2/(mu0_const*2*pi*frq*5.98e7[S/m]))`	2.657E-5 m	

COMPONENT 1 (COMP1)

1 In the **Model Builder** window, click **Component 1 (comp1)**.

2 In the **Settings** window for **Component**, locate the **Curved Mesh Elements** section.

3 From the **Geometry shape function** list, choose **Linear Lagrange**.

4 Clear the **Avoid inverted elements by curving interior domain elements** check box.

GEOMETRY 1

Insert the geometry sequence from the `resonant_spiral_coil_3d_geom_sequence.mph` file.

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `resonant_spiral_coil_3d_geom_sequence.mph`.

3 In the **Geometry** toolbar, click **Build All**.

DEFINITIONS

Copper Surface

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Selections>Explicit**.

3 In the **Settings** window for **Explicit**, type `Copper Surface` in the **Label** text field.

4 Locate the **Input Entities** section. Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `4-8` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Explicit**, locate the **Output Entities** section.

8 From the **Output entities** list, choose **Adjacent boundaries**.

Copper Domains

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Copper Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `4-8` in the **Selection** text field.

5 Click **OK**.

Contact Boundaries

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Contact Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `32` in the **Selection** text field.

6 Click **OK**.

Copper and Contact Boundaries

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Copper and Contact Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, select **Contact Boundaries** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Union**, locate the **Input Entities** section.

8 Under **Selections to add**, click **Add**.

9 In the **Add** dialog box, select **Copper Surface** in the **Selections to add** list.

10 Click **OK**.

Copper Edges

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Copper Edges` in the **Label** text field.
3 Locate the **Input Entities** section. Click **Paste Selection**.
4 In the **Paste Selection** dialog box, type `4-8` in the **Selection** text field.
5 Click **OK**.
6 In the **Settings** window for **Explicit**, locate the **Output Entities** section.
7 From the **Output entities** list, choose **Adjacent edges**.

Feed and Ground

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Feed and Ground` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Click **Paste Selection**.
5 In the **Paste Selection** dialog box, type `20, 43` in the **Selection** text field.
6 Click **OK**.

Varnish Boundaries

1 In the **Definitions** toolbar, click **Difference**.
2 In the **Settings** window for **Difference**, type `Varnish Boundaries` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.
5 In the **Add** dialog box, select **Copper and Contact Boundaries** in the **Selections to add** list.
6 Click **OK**.
7 In the **Settings** window for **Difference**, locate the **Input Entities** section.
8 Under **Selections to subtract**, click **Add**.
9 In the **Add** dialog box, select **Feed and Ground** in the **Selections to subtract** list.
10 Click **OK**.

View 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **View 1**.
2 In the **Settings** window for **View**, locate the **View** section.
3 Clear the **Show grid** check box.

ADD MATERIAL

1 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 Component** in the window toolbar.

MATERIALS

Copper (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
2 From the **Selection** list, choose **Copper Domains**.

The second material needed for the model is **Air**, used for the domain surrounding the coil. The material properties for air are readily available in COMSOL Multiphysics. They can be accessed using the **Add Material** window.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-in>Air**.
3 Click **Add to Component** in the window toolbar.

MATERIALS

Air (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
2 From the **Selection** list, choose **All domains**.
3 Drag and drop above **Copper (mat1)**.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **AC/DC>FR4 (Circuit Board)**.
3 Click **Add to Component** in the window toolbar.

MATERIALS

FR4 (Circuit Board) (mat3)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
2 Click **Paste Selection**.

3 In the **Paste Selection** dialog box, type 2 in the **Selection** text field.

4 Click **OK**.

5 In the **Settings** window for **Material**, locate the **Material Contents** section.

6 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0[S/m]	S/m	Basic

ADD MATERIAL

1 Go to the **Add Material** window.

2 Click **Add to Component** in the window toolbar.

3 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

FR4 (Circuit Board) 1 (mat4)

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

2 From the **Geometric entity level** list, choose **Boundary**.

3 From the **Selection** list, choose **Varnish Boundaries**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0[S/m]	S/m	Basic

MAGNETIC AND ELECTRIC FIELDS (MEF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic and Electric Fields (mef)**.

2 In the **Settings** window for **Magnetic and Electric Fields**, click to expand the **Discretization** section.

3 From the **Magnetic vector potential** list, choose **Linear**.

4 From the **Electric potential** list, choose **Linear**.

Magnetic Insulation 1

In the **Model Builder** window, under **Component 1 (comp1)> Magnetic and Electric Fields (mef)** click **Magnetic Insulation 1**.

Electric Insulation 1

1 In the **Physics** toolbar, click **Attributes** and choose **Electric Insulation**.

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

3 From the **Selection** list, choose **All boundaries**.

Magnetic Insulation 1

In the **Model Builder** window, click **Magnetic Insulation 1**.

Terminal 1

1 In the **Physics** toolbar, click **Attributes** and choose **Terminal**.

2 In the **Settings** window for **Terminal**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `20` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Terminal**, locate the **Terminal** section.

7 From the **Terminal type** list, choose **Voltage**.

8 In the V_0 text field, type `5[mV]`.

Magnetic Insulation 1

In the **Model Builder** window, click **Magnetic Insulation 1**.

Ground 2

1 In the **Physics** toolbar, click **Attributes** and choose **Ground**.

2 In the **Settings** window for **Ground**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `43` in the **Selection** text field.

5 Click **OK**.

Magnetic Continuity 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Magnetic Continuity**.

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

3 From the **Selection** list, choose **Varnish Boundaries**.

Contact Impedance 1

1 In the **Physics** toolbar, click **Attributes** and choose **Contact Impedance**.

2 In the **Settings** window for **Contact Impedance**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Varnish Boundaries**.

4 Locate the **Contact Impedance** section. In the d_s text field, type `0.0001[mm]`.

Magnetic Continuity 1

In the **Model Builder** window, click **Magnetic Continuity 1**.

Contact Impedance 2

1 In the **Physics** toolbar, click **Attributes** and choose **Contact Impedance**.

2 In the **Settings** window for **Contact Impedance**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Contact Boundaries**.

4 Locate the **Contact Impedance** section. In the d_s text field, type `0.0002[mm]`.

Note that two different Contact Impedance subfeatures are added to the Magnetic Continuity feature, since the varnish layer is twice as thick on the inner boundaries where the coil is in contact with itself.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **Size**.

2 Drag and drop **Size 1** below **Size**.

3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Edge**.

5 From the **Selection** list, choose **Copper Edges**.

6 Locate the **Element Size** section. Click the **Custom** button.

7 Locate the **Element Size Parameters** section.

8 Select the **Maximum element size** check box. In the associated text field, type `0.2`.

9 Click **Build Selected**.

Size 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Copper Domains**.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `0.5`.

8 Click **Build Selected**.

Now, add a Boundary Layer mesh. This is needed in order to resolve the small skin depth in the varnish layer without making the model too computationally heavy.

Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Copper Domains**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Copper and Contact Boundaries**.

4 Locate the **Layers** section. In the **Number of layers** text field, type `4`.

5 In the **Stretching factor** text field, type `1.5`.

6 From the **Thickness specification** list, choose **First layer**.

7 In the **Thickness** text field, type `delta/2`.

8 Click **Build Selected**.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.

2 In the **Settings** window for **Free Tetrahedral**, click to expand the **Element Quality Optimization** section.

3 Click **Build Selected**.

4 In the **Mesh** toolbar, click **Plot**.

RESULTS

Mesh Plot

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

2 In the **Settings** window for **3D Plot Group**, type `Mesh Plot` in the **Label** text field.

3 Click to expand the **Selection** section.

Mesh 1

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

2 In the **Settings** window for **Mesh**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Traffic>TrafficLightClassic** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Mesh**, locate the **Coloring and Style** section.

7 From the **Color table transformation** list, choose **Reverse**.

8 Click to expand the **Element Filter** section. Select the **Enable filter** check box.

9 In the **Expression** text field, type `y>1[mm]`.

Selection 1

1 Right-click **Mesh 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Copper Domains**.

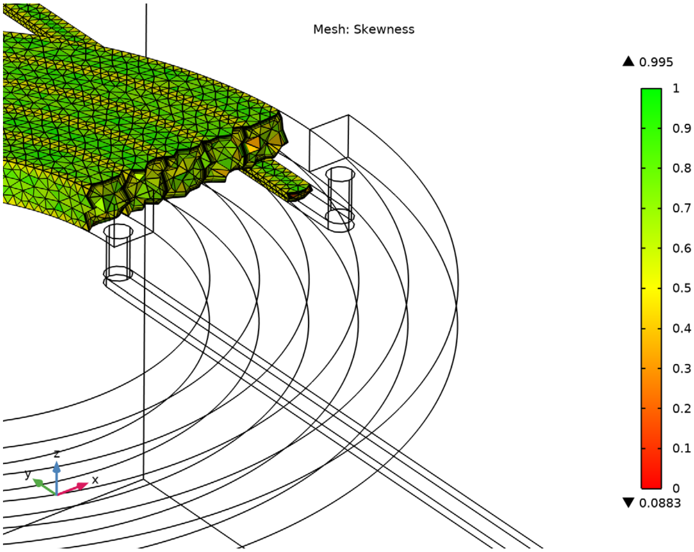
Mesh Plot

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

2 In the **Mesh Plot** toolbar, click **Plot**.

3 Click the **Go to Default View** button in the **Graphics** toolbar.

The plot shows the Mesh.



The last step consists of setting up the study. Since the electric potential converges rather slowly, the stationary tolerance needs to be tightened a bit in order to get accurate results. Then, set up an iterative solver.

STUDY 1

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

Step 1: Frequency Domain

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

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

3 In the **Frequencies** text field, type `range(5600000,28000,7000000)`.

4 From the **Reuse solution from previous step** list, choose **Auto**.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Stationary Solver 1**.

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1e-9`.

5 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node, then click **Iterative 1**.

6 In the **Settings** window for **Iterative**, locate the **General** section.

7 From the **Solver** list, choose **BiCGStab**.

8 From the **Preconditioning** list, choose **Right**.

9 Click to expand the **Error** section. In the **Factor in error estimate** text field, type `400`.

10 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1> Iterative 1** and choose **Direct Preconditioner**.

11 In the **Settings** window for **Direct Preconditioner**, locate the **General** section.

12 From the **Solver** list, choose **PARDISO**.

13 Click to expand the **Hybridization** section. From the **Use as** list, choose **Multi preconditioner**.

14 In the **Preconditioner variables** list, select **Magnetic vector potential (comp1.A)**.

15 Under **Preconditioner variables**, click **Delete**.

16 Right-click **Iterative 1** and choose **SOR**.

17 In the **Settings** window for **SOR**, locate the **General** section.

18 In the **Number of iterations** text field, type `3`.

19 Click to expand the **Hybridization** section. In the **Preconditioner variables** list, select **Electric potential (comp1.V)**.

20 Under **Preconditioner variables**, click **Delete**.

DEFINITIONS

Global Variable Probe 1 (var1)

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, locate the **Expression** section.

3 In the **Expression** text field, type `real(1/mef.Y11)`.

Global Variable Probe 2 (var2)

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, locate the **Expression** section.

3 In the **Expression** text field, type `imag(1/mef.Y11)`.

STUDY 1

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

RESULTS

B, J and V

1. In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2. In the **Settings** window for **3D Plot Group**, type `B, J and V` in the **Label** text field.
3. Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **6.188E6**.
4. Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Slice 1

1. Right-click **B, J and V** and choose **Slice**.
2. In the **Settings** window for **Slice**, locate the **Expression** section.
3. In the **Expression** text field, type `log(mef.normJ)`.
4. Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.
5. From the **Entry method** list, choose **Coordinates**.
6. In the **y-coordinates** text field, type `-2`.
7. Locate the **Coloring and Style** section. Click **Change Color Table**.
8. In the **Color Table** dialog box, select **Rainbow>Prism** in the tree.
9. Click **OK**.

Selection 1

1. Right-click **Slice 1** and choose **Selection**.
2. In the **Settings** window for **Selection**, locate the **Selection** section.
3. Click **Paste Selection**.
4. In the **Paste Selection** dialog box, type `4` in the **Selection** text field.
5. Click **OK**.

Arrow Surface 1

1. In the **Model Builder** window, right-click **B, J and V** and choose **Arrow Surface**.
2. In the **Settings** window for **Arrow Surface**, locate the **Expression** section.
3. In the **x-component** text field, type `side(4,mef.Jx)`.
4. In the **y-component** text field, type `side(4,mef.Jy)`.
5. In the **z-component** text field, type `side(4,mef.Jz)`.
6. Locate the **Arrow Positioning** section. From the **Placement** list, choose **Gauss points**.

7. Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.
8. From the **Arrow length** list, choose **Logarithmic**.
9. In the **Range quotient** text field, type `25`.
10. From the **Arrow base** list, choose **Center**.

Selection 1

1. Right-click **Arrow Surface 1** and choose **Selection**.
2. In the **Settings** window for **Selection**, locate the **Selection** section.
3. From the **Selection** list, choose **Copper Surface**.

Color Expression 1

1. In the **Model Builder** window, right-click **Arrow Surface 1** and choose **Color Expression**.
2. In the **Settings** window for **Color Expression**, locate the **Expression** section.
3. In the **Expression** text field, type `mef.normJ`.
4. In the **Unit** field, type `A/mm^2`.
5. Locate the **Coloring and Style** section. Click **Change Color Table**.
6. In the **Color Table** dialog box, select **Linear>GrayScale** in the tree.
7. Click **OK**.
8. In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.
9. Clear the **Color legend** check box.

Line 1

1. In the **Model Builder** window, right-click **B, J and V** and choose **Line**.
2. In the **Settings** window for **Line**, locate the **Expression** section.
3. In the **Expression** text field, type `1`.
4. Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
5. From the **Color** list, choose **Gray**.

Selection 1

1. Right-click **Line 1** and choose **Selection**.
2. In the **Settings** window for **Selection**, locate the **Selection** section.
3. From the **Selection** list, choose **Copper Edges**.

Volume 1

1. In the **Model Builder** window, right-click **B, J and V** and choose **Volume**.
2. In the **Settings** window for **Volume**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Rainbow>Dipole** in the tree.

5 Click **OK**.

Selection 1

1 Right-click **Volume 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `2` in the **Selection** text field.

5 Click **OK**.

Transparency 1

1 In the **Model Builder** window, right-click **Volume 1** and choose **Transparency**.

2 In the **Settings** window for **Transparency**, locate the **Transparency** section.

3 In the **Fresnel transmittance** text field, type `0.5`.

Arrow Volume 1

1 In the **Model Builder** window, right-click **B, J and V** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, locate the **Arrow Positioning** section.

3 Find the **x grid points** subsection. In the **Points** text field, type `40`.

4 Find the **y grid points** subsection. In the **Points** text field, type `1`.

5 Find the **z grid points** subsection. In the **Points** text field, type `40`.

6 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

7 From the **Arrow base** list, choose **Center**.

Color Expression 1

1 Right-click **Arrow Volume 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Expression** section.

3 In the **Expression** text field, type `mef.normB`.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Rainbow>PrismDark** in the tree.

6 Click **OK**.

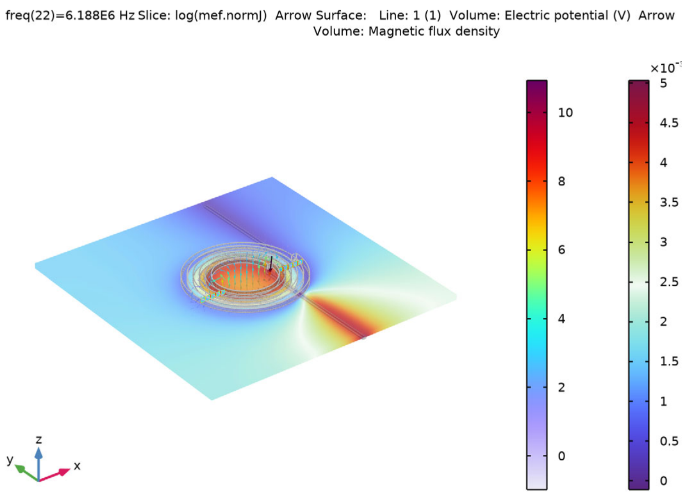
7 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

8 Clear the **Color legend** check box.

B, J and V

1. In the **Model Builder** window, under **Results** click **B, J and V**.
2. In the **B, J and V** toolbar, click **Plot**.
3. Click the **Scene Light** button in the **Graphics** toolbar.
4. Click the **Go to Default View** button in the **Graphics** toolbar.

   The Arrow Volume shows the Magnetic flux density. The volume shows the Electric potential. The Arrow Surface shows the Current density.



Normal (displacement) current density

1. In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2. In the **Settings** window for **3D Plot Group**, type `Normal (displacement) current density` in the **Label** text field.
3. Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **6.188E6**.
4. Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

1. Right-click **Normal (displacement) current density** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `abs(mef.mc1.ci2.nJ)`.
4. In the **Unit** field, type `A/mm^2`.
5. Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Rainbow>RainbowLightClassic** in the tree.

7 Click **OK**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Contact Boundaries**.

Line 1

1 In the **Model Builder** window, right-click **Normal (displacement) current density** and choose **Line**.

2 In the **Settings** window for **Line**, locate the **Expression** section.

3 In the **Expression** text field, type 1.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

Selection 1

1 Right-click **Line 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Copper Edges**.

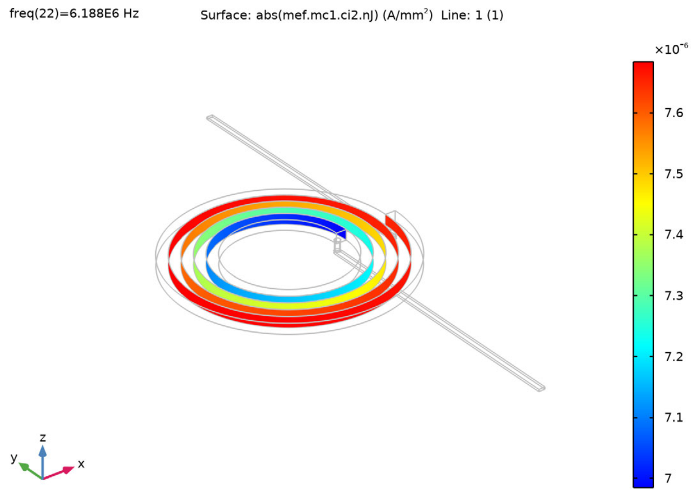
Normal (displacement) current density

1 In the **Model Builder** window, under **Results** click **Normal (displacement) current density**.

2 In the **Normal (displacement) current density** toolbar, click **Plot**.

3 Click the **Go to Default View** button in the **Graphics** toolbar.

The plot shows the Normal current density.



|J|

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type |J| in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **6.188E6**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Line 1

1 Right-click **|J|** and choose **Line**.

2 In the **Settings** window for **Line**, locate the **Expression** section.

3 In the **Expression** text field, type 1.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

Selection 1

1 Right-click **Line 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Copper Edges**.

Volume 1

1 In the **Model Builder** window, right-click **|J|** and choose **Volume**.

2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 In the **Expression** text field, type `mef.normJ`.

4 In the **Unit** field, type `A/mm^2`.

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Thermal>HeatCameraLight** in the tree.

7 Click **OK**.

8 In the **Settings** window for **Volume**, locate the **Coloring and Style** section.

9 From the **Color table transformation** list, choose **Reverse**.

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

Selection 1

1 Right-click **Volume 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Copper Domains**.

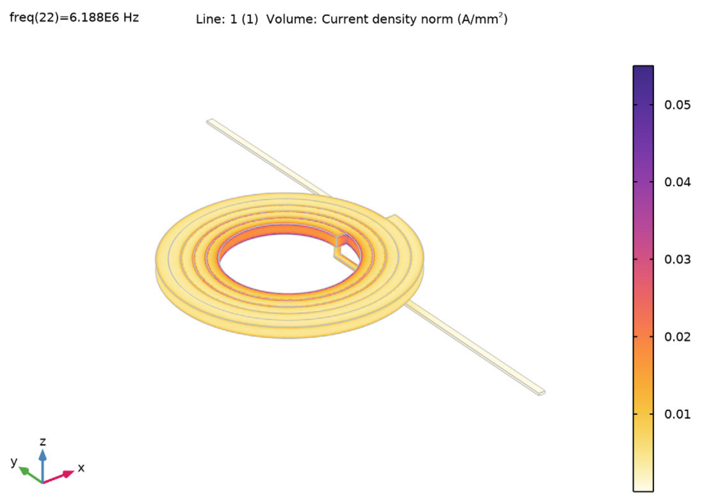
|J|

1 In the **Model Builder** window, under **Results** click **|J|**.

2 In the **|J|** toolbar, click **Plot**.

3 Click the Go to Default View button in the **Graphics** toolbar.

The Volume plot shows the norm of the current density.



Feed Impedance

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Feed Impedance` in the **Label** text field.

Global 1

1 Right-click **Feed Impedance** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
real(1/mef.Y11)	Ω	R
imag(1/mef.Y11)	Ω	X

4 Click to expand the **Coloring and Style** section. From the **Width** list, choose **3**.

Feed Impedance

1 In the **Model Builder** window, click **Feed Impedance**.

2 In the **Feed Impedance** toolbar, click **Plot**.