AC/DC Module

Application Library Manual

5.2a

COMSOL

AC/DC Module Application Library Manual

Version: COMSOL 5.2a

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM020102

Created in COMSOL Multiphysics 5.2a

Computing the Effect of Fringing Fields on Capacitance

Introduction

A typical capacitor is composed of two conductive objects with a dielectric in between them. Applying a potential difference between these objects results in an electric field. This electric field exists not just directly between the conductive objects, but extends some distance away, a phenomenon known as a fringing field. To accurately predict the capacitance of a capacitor, the domain used to model the fringing field must be sufficiently large, and the appropriate boundary conditions must be used. This example models a parallel plate capacitor in air and studies the size of the air domain. The choice of boundary condition is also addressed.

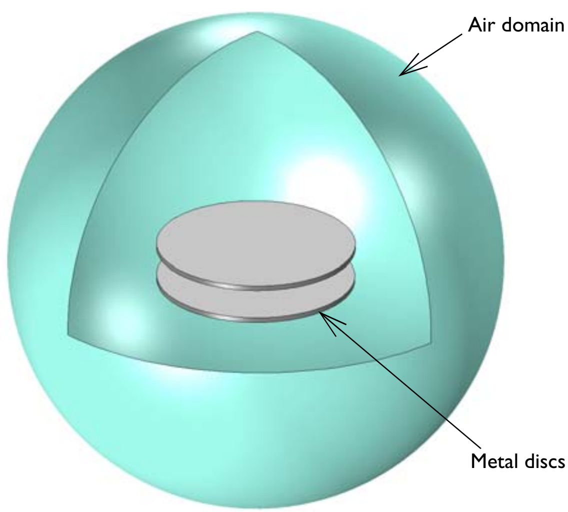


Figure 1: A simple capacitor consisting of two metal discs in an air domain.

Model Definition

Figure 1 shows the capacitor consisting of two metal discs in a spherical volume of air. The size of the sphere truncates the modeling space. This model studies the size of this air domain and its effect upon the capacitance.

One of the plates is specified as ground, with the electric potential 0 V. The other plate has an applied potential of 1 V.

The air sphere boundary can be thought of as one of two different physical situations: It can be treated as a perfectly insulating surface, across which charge cannot redistribute itself, or as a perfectly conducting surface, over which the potential does not vary.

The modeling realization of the perfectly insulating surface is the Zero Charge boundary condition. This boundary condition also implies that the electric field lines are tangential to the boundary.

The modeling realization of the perfectly conducting surface is the Floating Potential boundary condition. This boundary condition fixes the electric potential on all of the boundaries of the sphere to a constant, but unknown, value that is computed during the solution. The boundary condition also implies that the electric field lines are perpendicular to the boundary.

When studying convergence of results with respect to the surrounding domain size, it is important to fix the element size. In this model, the element size is fixed as the domain size is varied.

Results and Discussion

Figure 2 and Figure 3 plot the electric field for the cases where the air sphere boundary is treated as perfectly insulating and perfectly conducting, respectively. The fields terminate differently on the boundaries of the air sphere.

Figure 4 compares the capacitance values of the device with respect to air sphere radius for the two boundary conditions. The figure also plots the average of the two values. Notice that all three capacitance calculations converge to the same value as the radius grows. In practice, it is often sufficient to model a small air sphere with the electric insulation and floating potential boundary conditions and to take the average of the two.

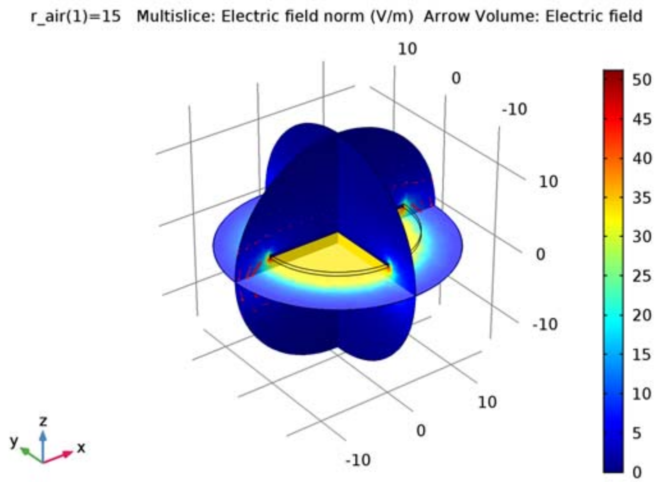


Figure 2: The electric field norm (multislices) and electric field (arrows) for the case of the Zero Charge boundary condition.

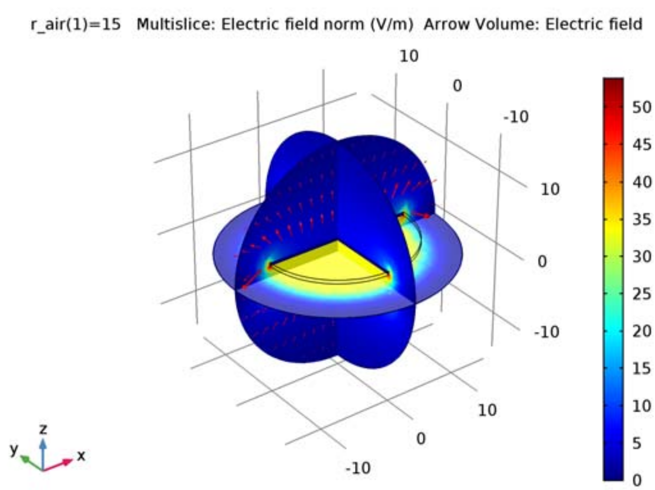


Figure 3: The electric field norm (multislices) and electric field (arrows) for the case of the Floating Potential boundary condition.

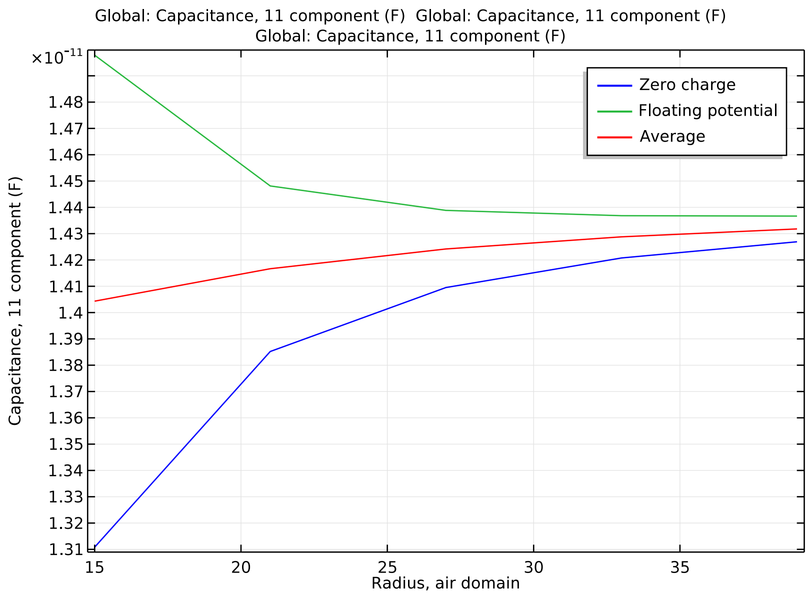


Figure 4: Convergence of the device capacitance as the size of the surrounding air sphere is increased. Electric insulation and fixed potential boundary conditions converge to the same result. The average of the two is also plotted.

Application Library path: `ACDC_Module/Capacitive_Devices/capacitor_fringing_fields`

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>Electrostatics (es)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
r_air	15[cm]	0.15 m	Radius, air domain

GEOMETRY 1

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

2 In the **Settings** window for Geometry, locate the **Units** section.

3 From the **Length unit** list, choose **cm**.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type 10.

4 In the **Height** text field, type 0.5.

5 Locate the **Position** section. In the **z** text field, type -2.

Mirror 1 (mir1)

1 Right-click **Cylinder 1 (cyl1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

3 Select the object **cyl1** only.

4 In the **Settings** window for Mirror, locate the **Input** section.

5 Select the **Keep input objects** check box.

Sphere 1 (sph1)

1 Right-click **Mirror 1 (mir1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Sphere**.

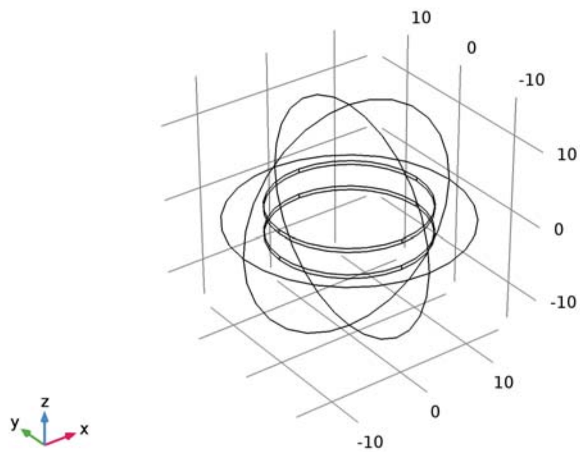
3 In the **Settings** window for Sphere, locate the **Size** section.

4 In the **Radius** text field, type `r_air`.

5 Right-click **Sphere 1 (sph1)** and choose **Build Selected**.

6 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

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



The geometry describes two metal discs in an air domain.

DEFINITIONS

Create a set of selections to use when setting up the physics. Begin with a selection for the outermost air domain boundaries.

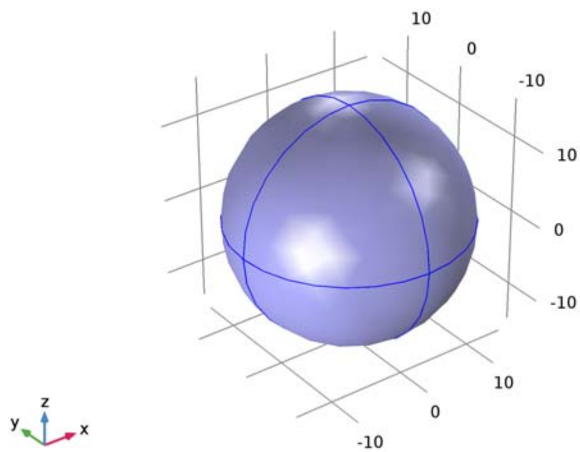
Explicit 1

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

2 In the **Settings** window for Explicit, locate the **Input Entities** section.

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

4 Select Boundaries 1–4, 13, 14, 17, and 18 only.



5 Right-click **Explicit 1** and choose **Rename**.

6 In the **Rename Explicit** dialog box, type `Outermost surface` in the **New label** text field.

7 Click **OK**.

View 1

Hide one boundary to get a better view of the interior parts when setting up the physics and reviewing the mesh.

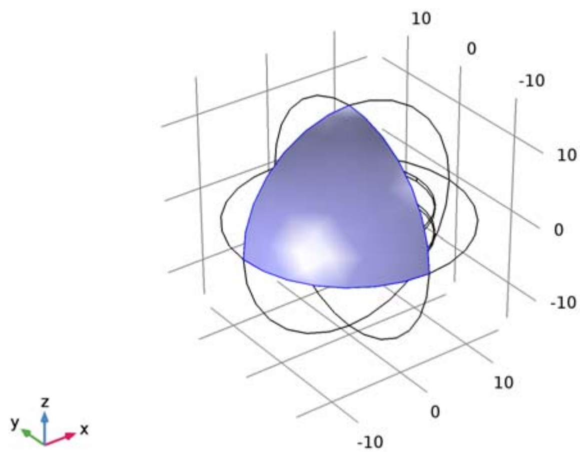
Hide for Physics 1

1 On the **View 1** toolbar, click **Hide for Physics**.

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

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

4 Select Boundary 2 only.



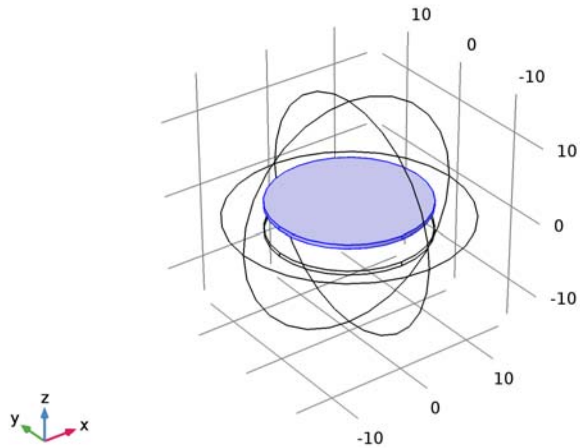
ELECTROSTATICS (ES)

The default boundary condition is **Zero Charge**, which is applied to all exterior boundaries. Add two **Terminal** features on the two electrodes, one connected to the source, and one connected to ground.

Terminal 1

1 On the **Physics** toolbar, click **Domains** and choose **Terminal**.

2 Select Domain 3 only.



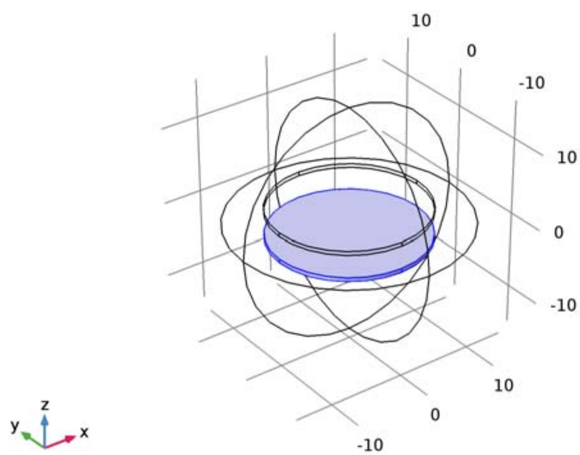
3 In the **Settings** window for Terminal, locate the **Terminal** section.

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

Terminal 2

1 On the **Physics** toolbar, click **Domains** and choose **Terminal**.

2 Select Domain 2 only.



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

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

5 In the V_0 text field, type 0.

MATERIALS

Next, assign material properties on the model. Specify air for all domains.

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MESH 1

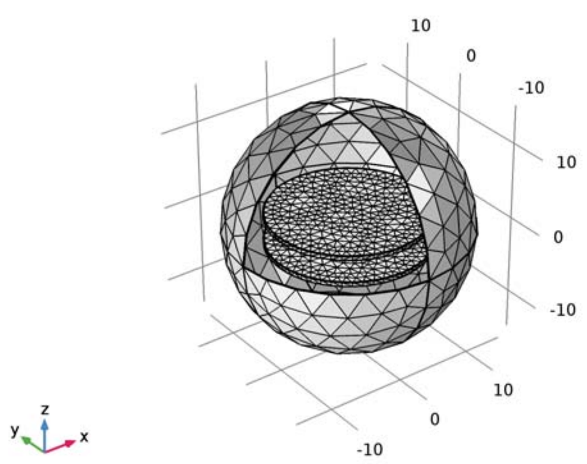
In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

Size

1 In the **Settings** window for Size, locate the **Element Size** section.

2 From the **Predefined** list, choose **Coarse**.

3 Click **Build All**.



STUDY 1

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
r_air	range(15,6,39)	

5 On the **Study** toolbar, click **Compute**.

RESULTS

Electric Potential (es)

Modify the default plot to show the electric field norm. Add an arrow plot for the electric field to observe the field direction.

1 In the **Model Builder** window, under **Results** click **Electric Potential (es)**.

2 In the **Settings** window for 3D Plot Group, locate the **Data** section.

3 From the **Parameter value (r_air)** list, choose **15**.

Multislice 1

1 In the **Model Builder** window, expand the **Electric Potential (es)** node, then click **Multislice 1**.

2 In the **Settings** window for Multislice, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Electrostatics> Electric>es.normE - Electric field norm**.

Electric Potential (es)

In the **Model Builder** window, under **Results** right-click **Electric Potential (es)** and choose **Arrow Volume**.

Arrow Volume 1

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

2 Find the **x grid points** subsection. In the **Points** text field, type 20.

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

4 Find the **z grid points** subsection. In the **Points** text field, type 10.

5 On the **Electric Potential (es)** toolbar, click **Plot**.

Compare the resulting plot with Figure 2.

Next, apply a **Floating Potential** boundary condition on the outermost surface. This condition overrides the default **Zero Charge** condition.

ELECTROSTATICS (ES)

Floating Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Floating Potential**.

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

3 From the **Selection** list, choose **Outermost surface**.

Add a new study to keep the result from the previous computation.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
r_air	range(15,6,39)	

5 In the **Model Builder** window, click **Study 2**.

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

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

8 On the **Study** toolbar, click **Compute**.

RESULTS

Electric Potential (es)

In the **Model Builder** window, under **Results** right-click **Electric Potential (es)** and choose **Duplicate**.

Electric Potential (es) 1

1 In the **Settings** window for 3D Plot Group, locate the **Data** section.
2 From the **Data set** list, choose **Study 2/Parametric Solutions 2 (sol9)**.
3 On the **Electric Potential (es) 1** toolbar, click **Plot**.

   The reproduced plot should look like Figure 3.

Join 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Join**.
2 In the **Settings** window for Join, locate the **Data 1** section.
3 From the **Data** list, choose **Study 1/Parametric Solutions 1 (sol2)**.
4 Locate the **Data 2** section. From the **Data** list, choose **Study 2/Parametric Solutions 2 (sol9)**.
5 Locate the **Combination** section. From the **Method** list, choose **General**.
6 In the **Expression** text field, type `(data1+data2)/2`.

1D Plot Group 3

On the **Results** toolbar, click **1D Plot Group**.

Global 1

1 On the **1D Plot Group 3** toolbar, click **Global**.
2 In the **Settings** window for Global, locate the **Data** section.
3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.
4 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Electrostatics>Terminals>Capacitance>es.C11 - Capacitance, 11 component**.
5 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Model>Global Definitions>Parameters>r_air - Radius, air domain**.
6 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
Zero charge

8 On the **1D Plot Group 3** toolbar, click **Plot**.

Global 2

1 Right-click **Global 1** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Study 2/Parametric Solutions 2 (sol9)**.

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

Legends
Floating potential

5 On the **1D Plot Group 3** toolbar, click **Plot**.

Global 3

1 Right-click **Results>1D Plot Group 3>Global 2** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Join 1**.

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

Legends
Average

5 On the **1D Plot Group 3** toolbar, click **Plot**.

This reproduces Figure 4.

Optionally, to allow recomputing **Study 1**, you can disable the **Floating Potential** boundary condition for that study as follows.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

2 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Electrostatics (es)>Floating Potential 1**.

5 Click **Disable**.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)**.

Created in COMSOL Multiphysics 5.2a

Contact Impedance Comparison

Introduction

The contact impedance boundary condition is meant to approximate a thin layer of material that impedes the flow of current normal to the boundary, but does not introduce any additional conduction path tangential to the boundary. This example compares the contact impedance boundary condition to a full-fidelity model and discusses the range of applicability of this boundary condition.

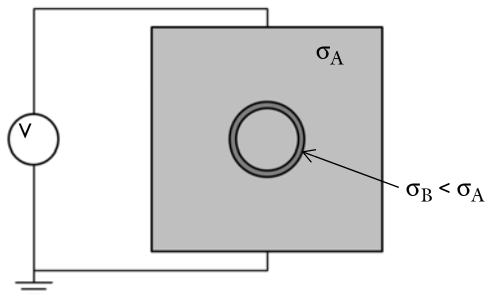


Figure 1: A square two-dimensional domain of conductive material, with a circular inclusion. The wall of the inclusion are made of a material with lower conductivity.

Model Definition

The situation being modeled is shown in Figure 1. A square two-dimensional domain of conductive material has a DC voltage difference applied to it. Within the square domain, there is a circular inclusion. The walls of this inclusion are modeled two ways: using a full fidelity model that includes the thickness of the walls, and using the contact impedance boundary condition. The interior of the inclusion has the same properties as the bulk.

The location of the contact impedance condition is at the centerline, midway between the inner and outer radii of the full fidelity model. Note that, when using the contact impedance boundary condition, the total volume of the surrounding material is slightly larger, since the thickness of the wall is not being explicitly modeled. The conductivity of the wall of the inclusion is varied between a value several orders of magnitude smaller to a value equal to the conductivity of the bulk.

Results and Discussion

The voltage distribution and the electric field strength is plotted in Figure 2 for the case where the electric conductivity is a thousand times smaller in the wall of the inclusion. This case represents a thin walled object that resists current flow through the surface, that is, it presents a high impedance to the voltage source. The field lines can be observed to deform around the object. The solutions agree well for the cases where the conductivity of the contact impedance boundary condition is less than the surrounding medium.

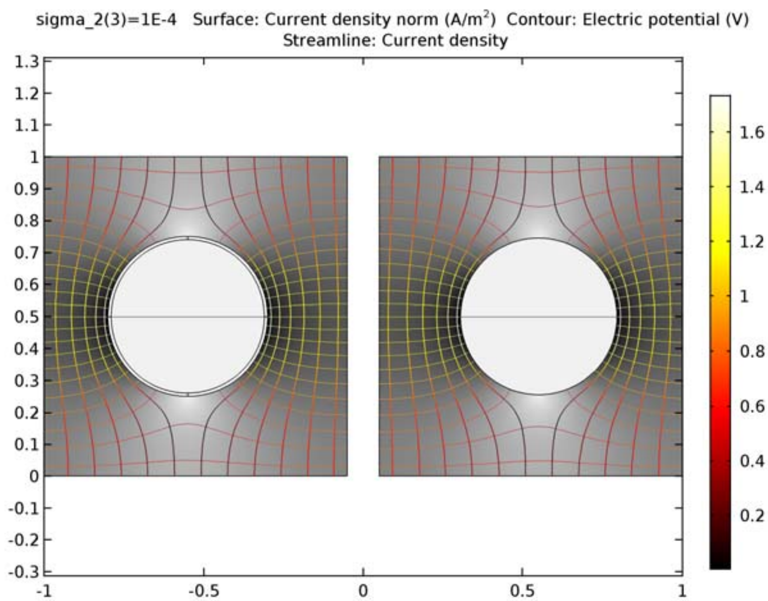


Figure 2: Isolines of the voltage field, electric field lines, and a greyscale plot of the current density for the case of a thin layer of material that has high impedance. The full fidelity and contact impedance solutions are almost identical.

The case of equal conductivities is plotted in Figure 3, and shows the limit of the contact impedance boundary condition. As the conductivities becomes equal, there can be no current flow tangential to the boundary.

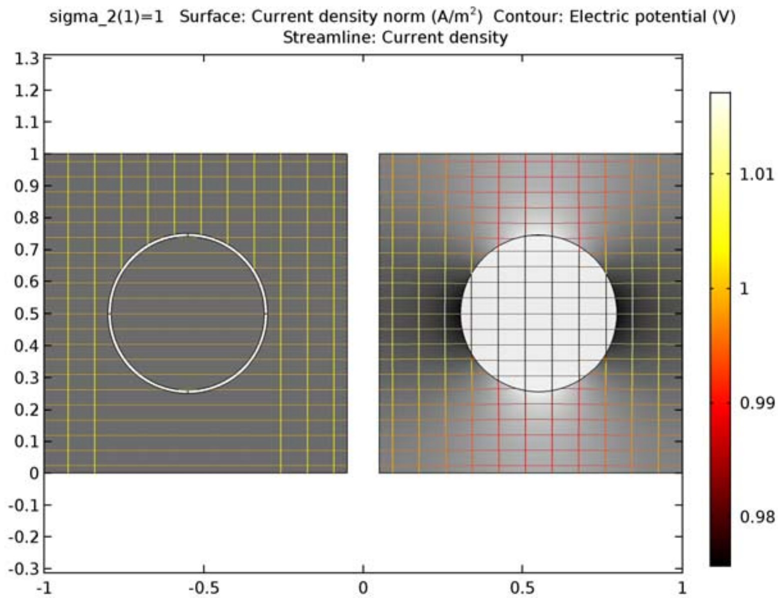


Figure 3: Isolines of the voltage field, electric field lines, and a grayscale plot of the current density. The contact impedance boundary condition (right) prevents any tangential current flow.

The Contact Impedance boundary condition can be used in cases where the thickness of the boundary being approximated is much smaller than the characteristic size of the model domain, and when the conductivity of the layer is smaller than the surrounding medium. When this boundary condition can be used, the resulting number of mesh element is much smaller, saving solution time and memory.

Application Library path: `ACDC_Module/Resistive_Devices/contact_impedance_comparison`

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 **2D**.

2 In the **Select Physics** tree, select **AC/DC>Electric Currents (ec)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
sigma_1	1[S/m]	1 S/m	Conductivity, material 1
sigma_2	1[S/m]	1 S/m	Conductivity, material 2

GEOMETRY 1

Square 1 (sq1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Square**.

2 In the **Settings** window for Square, locate the **Position** section.

3 In the **x** text field, type `0.05`.

Square 2 (sq2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Square**.

2 In the **Settings** window for Square, locate the **Position** section.

3 In the **x** text field, type `-1.05`.

4 Click **Build All Objects**.

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

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `0.245`.

4 Locate the **Position** section. In the **x** text field, type `0.55`.

5 In the **y** text field, type `0.5`.

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `0.25`.

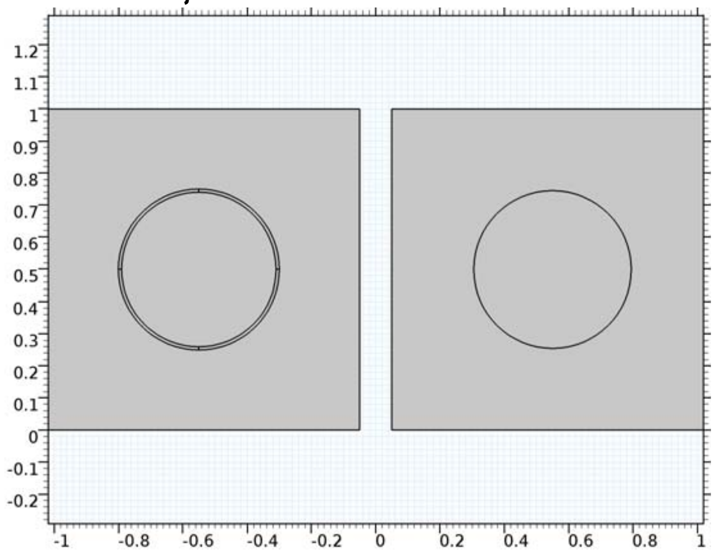
4 Locate the **Position** section. In the **x** text field, type `-0.55`.

5 In the **y** text field, type `0.5`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.01

7 Click **Build All Objects**.



The geometry on the left side describes the full fidelity model. The geometry on the right side replaces the thin layer with a boundary in order to use the **Contact Impedance** feature.

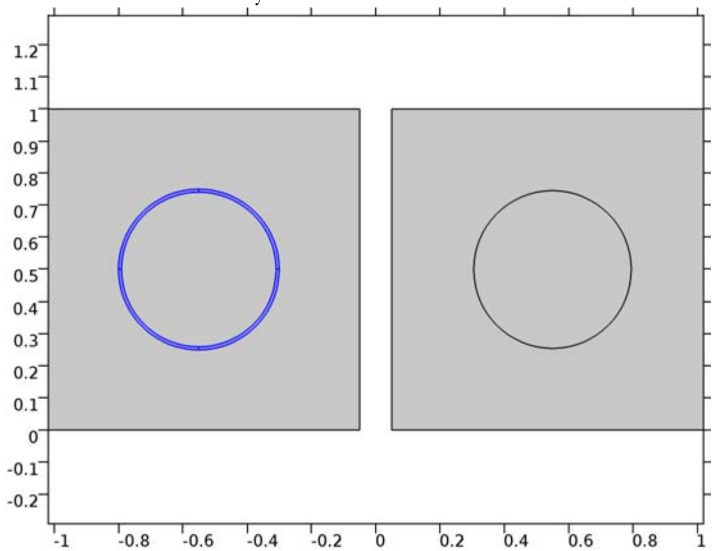
DEFINITIONS

Create a set of selections for use before setting up the physics. First, create a selection for the wall of the inclusion.

Explicit 1

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

2 Select Domains 2–5 only.



3 Right-click **Explicit 1** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Full fidelity` in the **New label** text field.

5 Click **OK**.

Add a selection for the contact impedance boundaries.

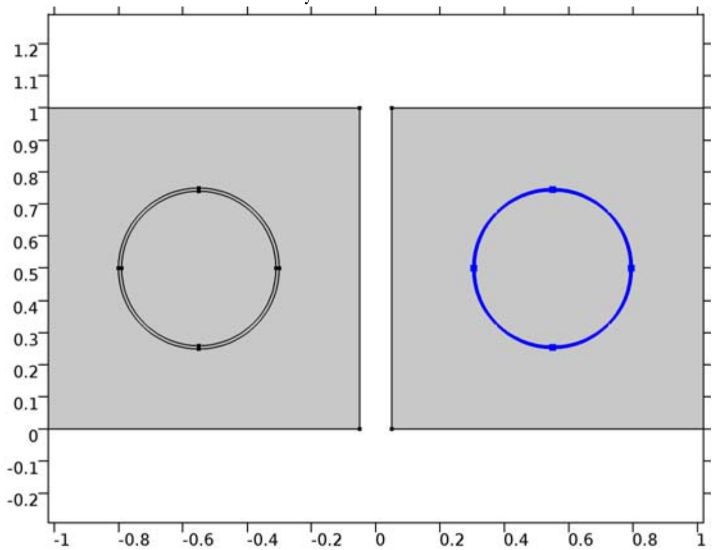
Explicit 2

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

2 In the **Settings** window for Explicit, locate the **Input Entities** section.

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

4 Select Boundaries 21–24 only.



5 Right-click **Explicit 2** and choose **Rename**.

6 In the **Rename Explicit** dialog box, type `Contact impedance` in the **New label** text field.

7 Click **OK**.

Add a selection for the bulk area. This is the complementary domain of the wall of the inclusion.

Complement 1

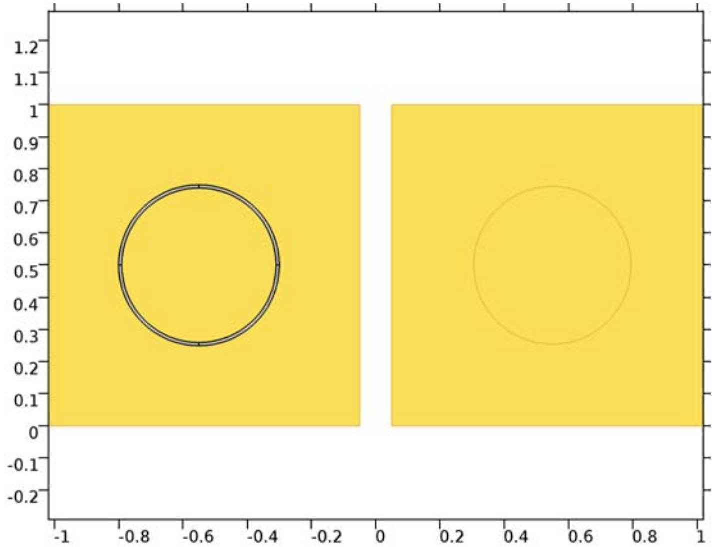
1 On the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for Complement, locate the **Input Entities** section.

3 Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **Full fidelity** in the **Selections to invert** list.

5 Click **OK**.



6 Right-click **Complement 1** and choose **Rename**.

7 In the **Rename Complement** dialog box, type `Bulk` in the **New label** text field.

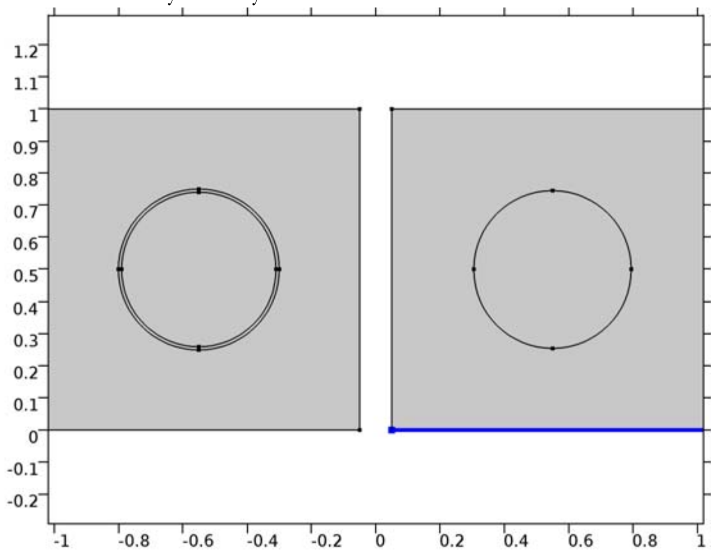
8 Click **OK**.

ELECTRIC CURRENTS (EC)

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

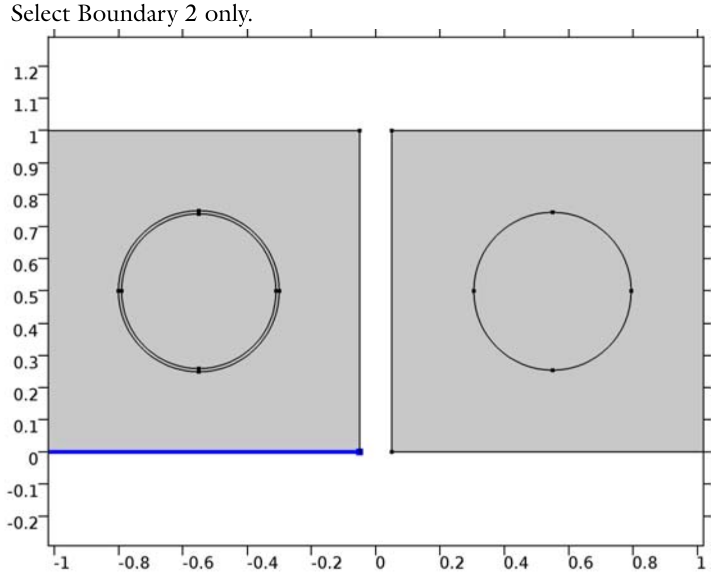
2 Select Boundary 10 only.



Ground 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

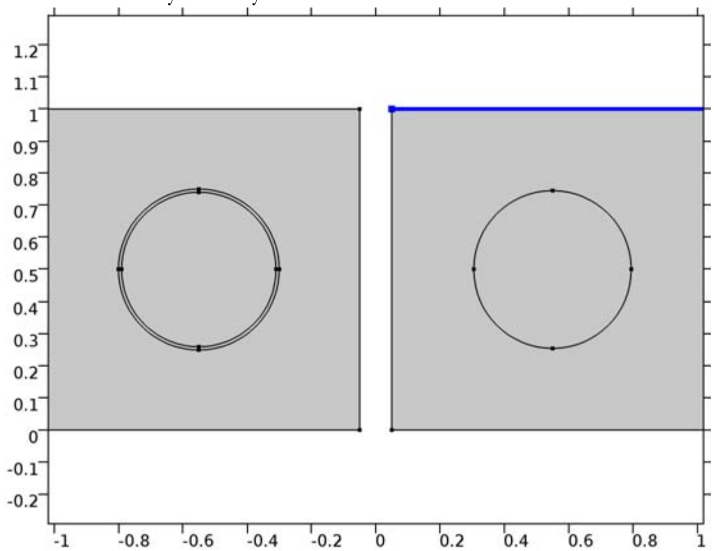
2 Select Boundary 2 only.



Terminal 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Terminal**.

2 Select Boundary 11 only.



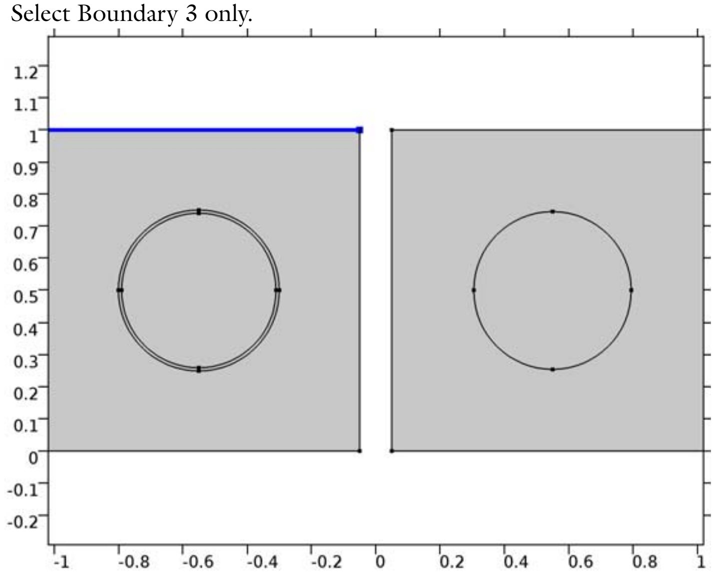
3 In the **Settings** window for Terminal, locate the **Terminal** section.

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

Terminal 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Terminal**.

2 Select Boundary 3 only.



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

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

Contact Impedance 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Contact Impedance**.

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

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

4 Locate the **Contact Impedance** section. In the d_s text field, type `1[cm]`.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

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

2 From the **Selection** list, choose **Bulk**.

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

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	sigma_1	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic

Material 2 (mat2)

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

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

3 From the **Selection** list, choose **Full fidelity**.

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

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	sigma_2	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic

Material 3 (mat3)

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

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

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

4 From the **Selection** list, choose **Contact impedance**.

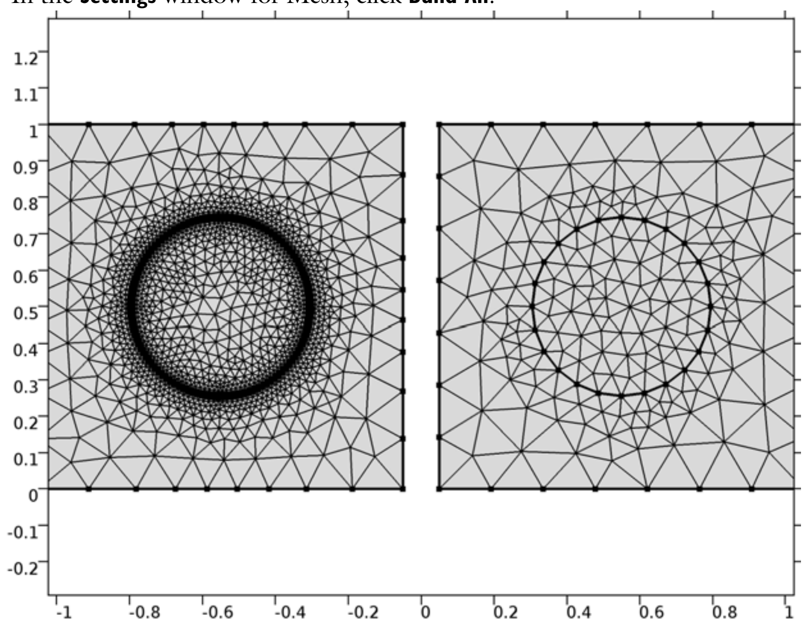
5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	`sigma_2`	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic

MESH 1

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

2 In the **Settings** window for Mesh, click **Build All**.



STUDY 1

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
sigma_2	1 0.01 0.0001	

5 On the **Study** toolbar, click **Compute**.

RESULTS

Electric Potential (ec)

Begin the result analysis by suppressing the domain of the wall of the inclusion which is not of interest.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/Solution 1 (sol1)**.

Selection

1. On the **Results** toolbar, click **Selection**.
2. In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. From the **Selection** list, choose **Bulk**.

   The default plot shows the surface plot of the electric potential. Change the expression to show the norm of the current density.

Surface 1

1. In the **Model Builder** window, expand the **Results>Electric Potential (ec)** node, then click **Surface 1**.
2. In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Electric Currents> Currents and charge>ec.normJ - Current density norm**.
3. Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayPrint**.
4. Clear the **Color legend** check box.
5. Select the **Reverse color table** check box.

Electric Potential (ec)

Next, add a contour plot showing the electric potential.

Contour 1

1 In the **Model Builder** window, under **Results** right-click **Electric Potential (ec)** and choose **Contour**.

2 In the **Settings** window for Contour, locate the **Levels** section.

3 In the **Total levels** text field, type 21.

Color Expression 1

1 Right-click **Results>Electric Potential (ec)>Contour 1** and choose **Color Expression**.

2 In the **Settings** window for Color Expression, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Electric Currents>Currents and charge>ec.normJ - Current density norm**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalEquidistant**.

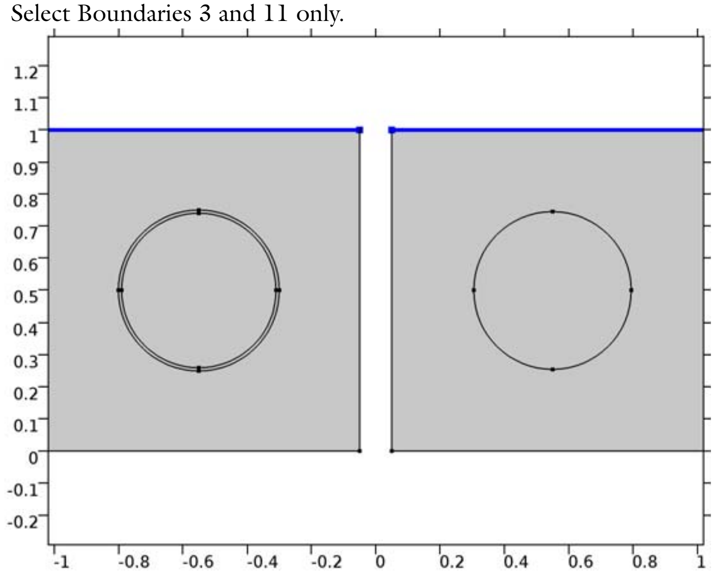
Electric Potential (ec)

Then, add a streamline plot of the current density.

Streamline 1

1 In the **Model Builder** window, under **Results** right-click **Electric Potential (ec)** and choose **Streamline**.

2 Select Boundaries 3 and 11 only.



3 In the **Settings** window for Streamline, locate the **Streamline Positioning** section.

4 In the **Number** text field, type 24.

Color Expression 1

1 Right-click **Results>Electric Potential (ec)>Streamline 1** and choose **Color Expression**.

2 In the **Settings** window for Color Expression, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Electric Currents>Currents and charge>ec.normJ - Current density norm**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalEquidistant**.

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

5 On the **Electric Potential (ec)** toolbar, click **Plot**.

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

Compare the plot with Figure 2.

Electric Potential (ec)

1 In the **Model Builder** window, under **Results** click **Electric Potential (ec)**.

2 In the **Settings** window for 2D Plot Group, locate the **Data** section.

3 From the **Parameter value (sigma_2)** list, choose **1**.

4 On the **Electric Potential (ec)** toolbar, click **Plot**.

This should look like Figure 3. Due to the selection defined in **Solution 1**, the streamlines in the full fidelity model are not completely visualized.

E-Core Transformer

Introduction

Transformers are electrical components that are used for power transmission. Most transformers work on the principle of electromagnetic induction. A typical transformer consists of a primary coil, a secondary coil and a ferromagnetic core. The primary coil receives the AC electrical input signal. As a result of mutual induction, an induced voltage is obtained across the secondary coil. The ferromagnetic core serves the purpose of a magnetic flux concentrator thereby minimizing losses due to flux leakage.

Commercial transformers use several types of cores, which are named based on their geometric shapes, for example I-core, U-core, E-core, pot core, toroidal, and planar. This application uses a pair of E-cores for magnetic flux concentration.

This example demonstrates how to perform transient simulations of a single-phase E-core transformer. Including the effect of a nonlinear B-H curve in the soft-iron core, the model computes the spatial distribution of the magnetic and electric fields, the magnetic saturation effect, the transient response, and the flux leakage to the surroundings. Two different versions of the transformer are simulated: the first one with a turn ratio of unity and the second one with a turn ratio of 1000.

Model Definition

The core of the single-phase E-core transformer considered here consists of a pair of E-cores, which form a closed magnetic flux path. The primary and secondary coils in the transformer are placed around the central leg of the core as shown in Figure 1.

A nonlinear B-H curve that includes saturation effects is used to simulate the magnetic behavior of the soft-iron core. Hysteresis effects in the core are neglected. The model assumes that the primary and secondary windings are made of thin wire and have multiple turns. Using the assumptions that the wire diameter is less than the skin depth and that there are many turns, these windings are modeled with Coil features. Furthermore, the model does not account for eddy currents in the individual turns of the coil. The primary winding is connected to a primary resistor, R_p and the AC voltage source, V_{ac} while the secondary winding is connected to the secondary load resistor, R_s as shown in Figure 2.

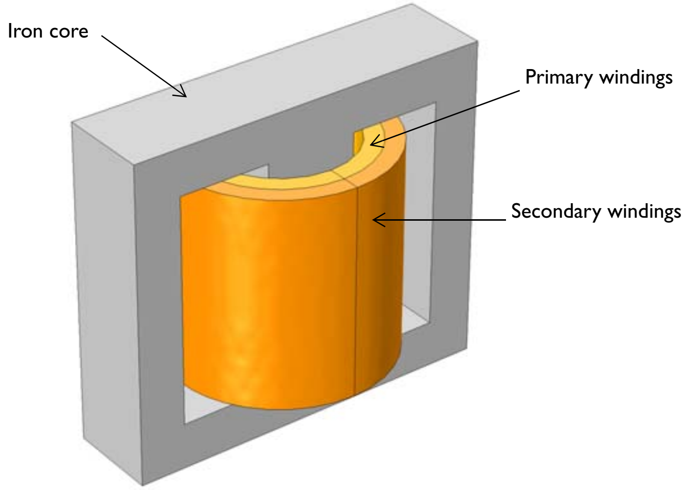


Figure 1: Model illustration of an E-core transformer.

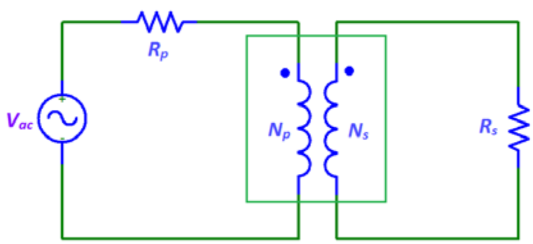


Figure 2: A transformer connected to an external circuit with voltage source and resistors.

The model is solved in time domain for a line frequency of 50 Hz. Several important design parameters such as the magnitude of the input voltage, the line frequency, the number of turns in the coils, and the coil resistance are parameterized and can therefore easily be changed.

A transformer works by the principle of Faraday's law of induction which states that the induced voltage (V_{in}) in a coil is proportional to the rate of change of magnetic flux (ϕ) and the number of turns (N) in a coil as shown in Equation 1.

$$V_{\text{in}} = -N\frac{d\phi}{dt} \tag{1}$$

If two coils are coupled, Equation 1 can be used to deduce that the induced voltage in the secondary coil (V_s) is proportional to the induced voltage in the primary coil (V_p):

$$\frac{V_s}{V_p} = \frac{N_s}{N_p} \tag{2}$$

Here N_s and N_p are the number of turns in the secondary and primary coils, respectively. N_p/N_s is known as the turn ratio.

Results and Discussion

Figure 3 shows the surface plot of the magnetic flux density norm distribution and the arrow plots of the currents in the windings at $t = 50$ ms.

Figure 4 shows the slice and the arrow plot of the magnetic flux density norm in the core at $t = 50$ ms.

Figure 5 and Figure 6 display the induced voltage in the primary and secondary windings respectively. Since the number of turns on each winding is equal, the induced voltage in both windings is same as given by Equation 2.

The currents flowing through the primary and secondary windings are shown in Figure 7 and Figure 8, respectively.

Figure 9 displays the voltage induced in the primary winding for a step-down transformer with a turn ratio of N_p/N_s = 1000 and supply voltage of 25 kV.

Finally, the induced voltage in the secondary winding for a step-down transformer is displayed in Figure 10. This induced voltage is 1000 times smaller compared to the voltage in the primary winding of Figure 9.

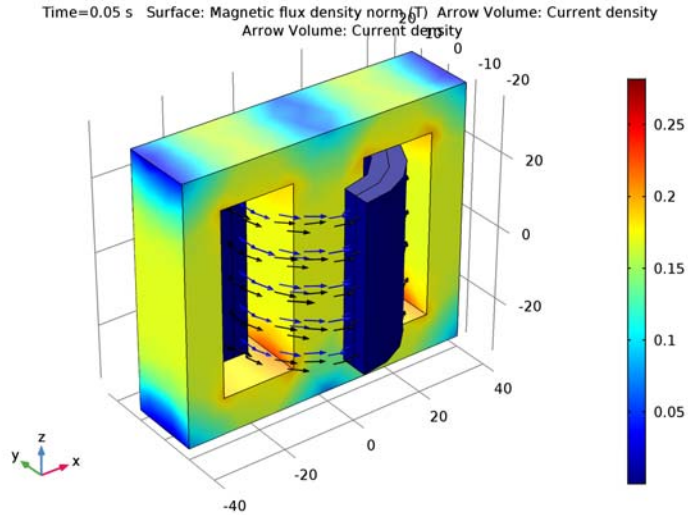


Figure 3: Magnetic flux density norm and the currents in the windings at t = 50ms.

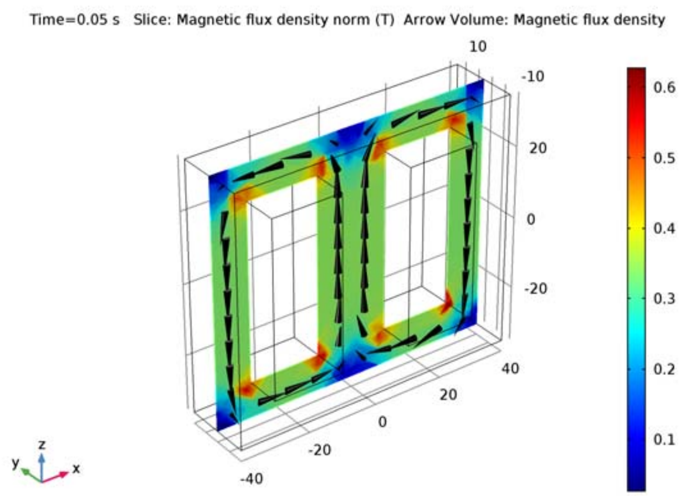


Figure 4: Magnetic flux density inside the transformer core at t = 50ms.

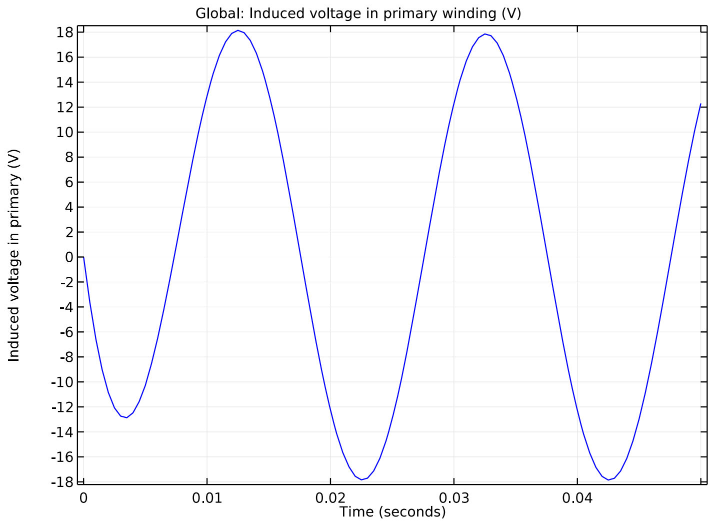


Figure 5: The induced voltage in the primary winding versus time.

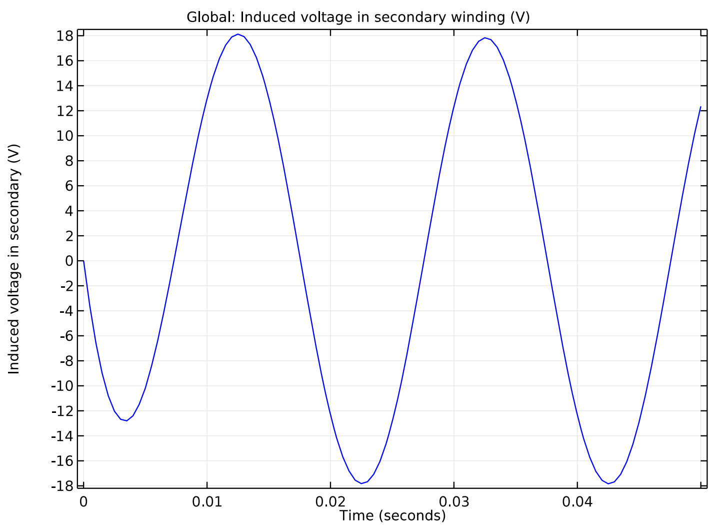


Figure 6: Induced voltage in the secondary windings versus time.

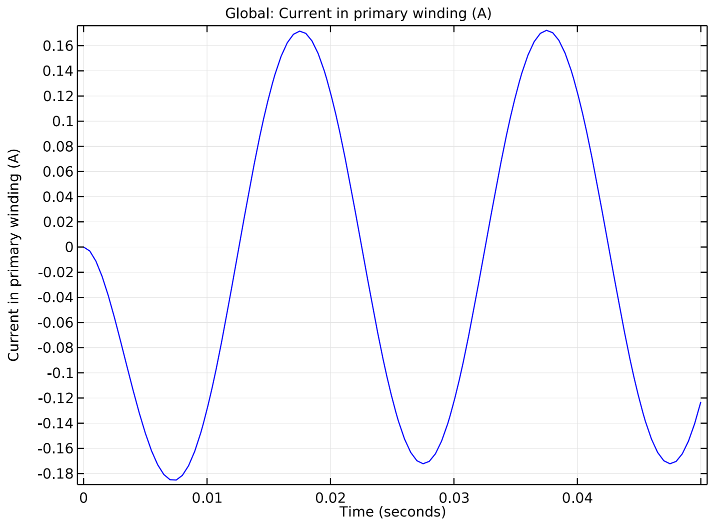


Figure 7: Current in the primary winding versus time.

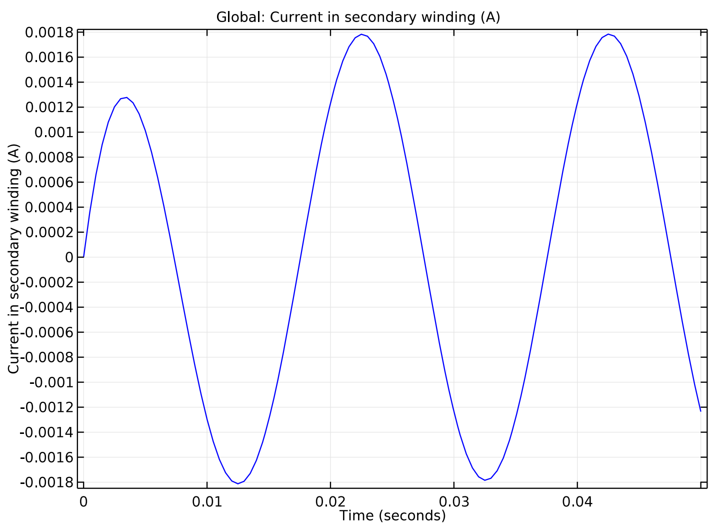


Figure 8: Current in the secondary winding versus time.

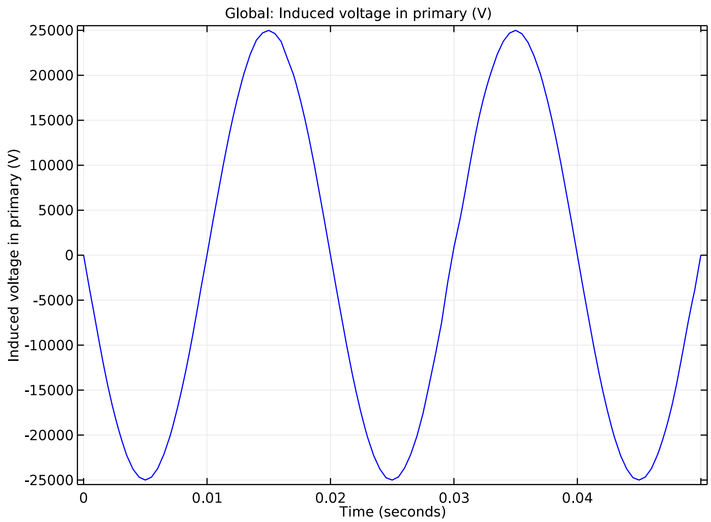


Figure 9: Induced voltage in the primary winding versus time for a step-down transformer.

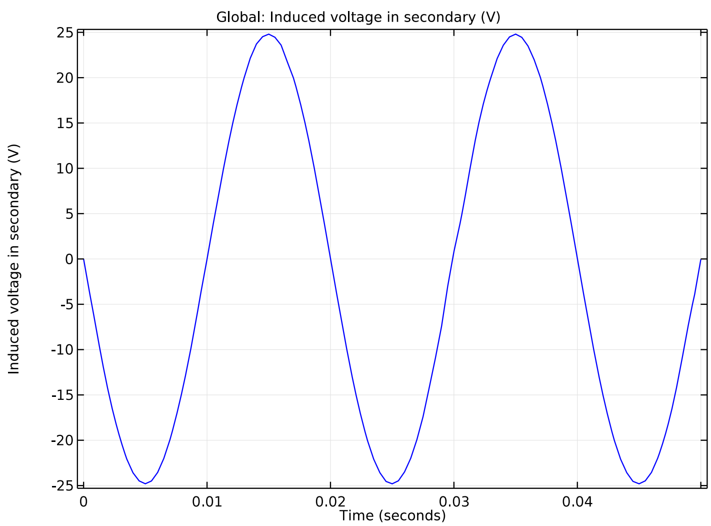


Figure 10: Induced voltage in the secondary winding versus time for a step-down transformer.

Notes About the COMSOL Implementation

Use the Magnetic Fields interface to model the magnetic fields of the transformer. Model the primary and secondary windings with Coil features. Connect the primary and secondary windings to an external circuit with the AC voltage source and resistors using an Electrical Circuit interface. Add a Coil Geometry Analysis study step to calculate the current in the coils. Perform a Time Dependent study to determine the voltage and currents in both the primary and secondary windings.

Application Library path: `ACDC_Module/Other_Industrial_Applications/ecore_transformer`

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>Magnetic Fields (mf)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **AC/DC>Electrical Circuit (cir)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Time Dependent**.
8 Click **Done**.

GLOBAL DEFINITIONS

Define all the required parameters.

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
Rp	100[ohm]	100 Ω	Primary side resistance
Rs	10[kohm]	1E4 Ω	Secondary side resistance
Np	300	300	Number of turns in primary winding
Ns	300	300	Number of turns in secondary winding
nu	50[Hz]	50 Hz	Frequency of supply voltage
Vac	25[V]	25 V	Supply voltage

GEOMETRY 1

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

1 On the **Geometry** toolbar, click **Insert Sequence**.

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

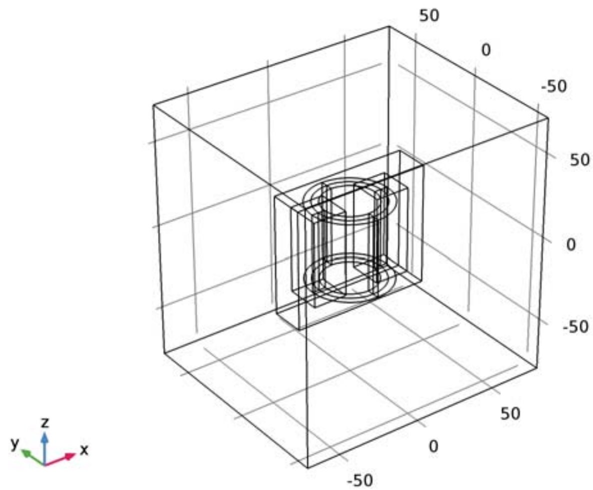
Form Union (fin)

1 On the **Geometry** toolbar, click **Build All**.

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

Choose wireframe rendering to get a better view of the interior parts.

3 Click the **Wireframe Rendering** button on the **Graphics** toolbar.



The geometry looks as shown in the figure above.

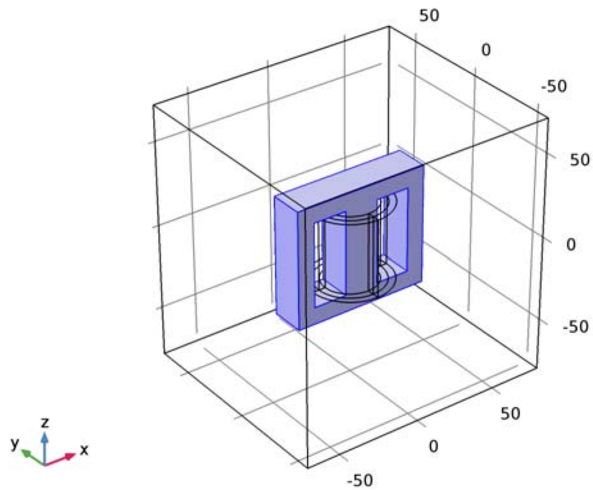
DEFINITIONS

Define domain and boundary selections for the core and the windings. First, create a selection for the core domain.

Explicit 1

1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Model Builder** window, right-click **Explicit 1** and choose **Rename**.
3. In the **Rename Explicit** dialog box, type `Core` in the **New label** text field.
4. Click **OK**.
5. Select Domain 2 only.

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



Specify a selection for the primary winding.

Explicit 2

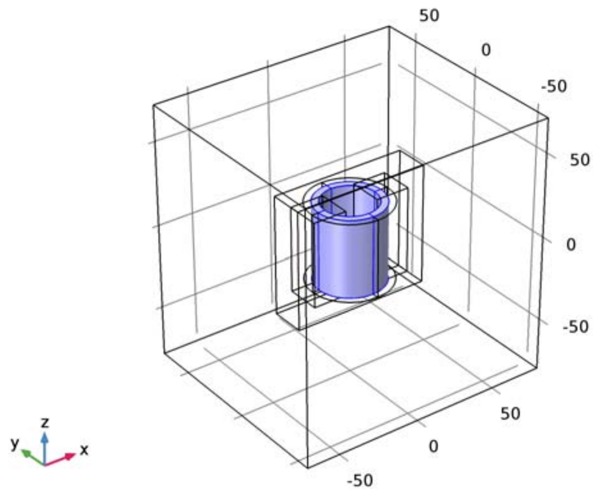
1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Model Builder** window, right-click **Explicit 2** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `Primary Winding` in the **New label** text field.

4 Click **OK**.

5 Select Domains 5, 6, 8, and 9 only.



Add a selection for the secondary winding.

Explicit 3

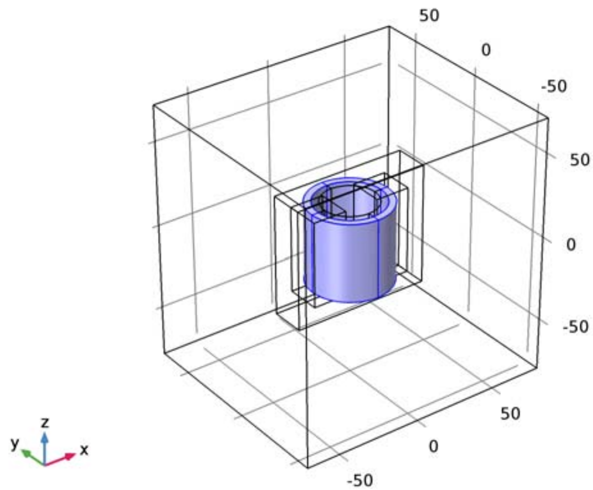
1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Model Builder** window, right-click **Explicit 3** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `Secondary Winding` in the **New label** text field.

4 Click **OK**.

5 Select Domains 3, 4, 7, and 10 only.



Explicit 4

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

2 In the **Model Builder** window, right-click **Explicit 4** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `Windings` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for Explicit, locate the **Input Entities** section.

6 Click **Paste Selection**.

7 In the **Paste Selection** dialog box, type `3-10` in the **Selection** text field.

8 Click **OK**.

Now set up the physics for the magnetic field. Start by hiding the outer boundaries to visualize the results only in the transformer core and windings.

DEFINITIONS

View 1

In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **View 1**.

Hide for Physics 1

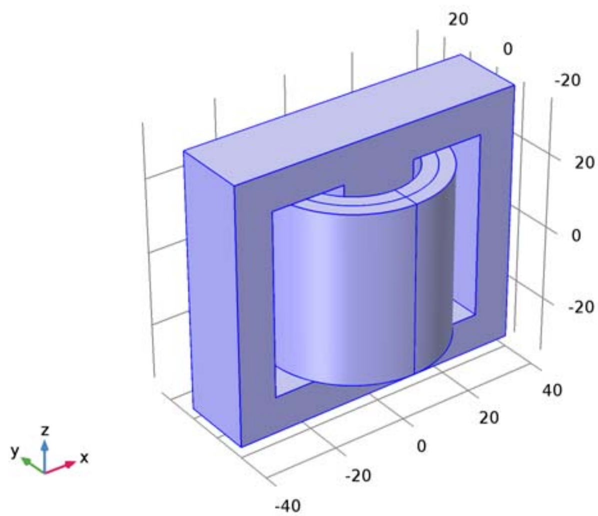
1 On the **View 1** toolbar, click **Hide for Physics**.

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

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

4 Select Boundaries 1–5 and 56 only.

MAGNETIC FIELDS (MF)



Choose the linear vector elements to discretize the magnetic vector potential. This will make the model computationally efficient. Typically, the default quadratic elements are recommended.

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

2 In the **Model Builder** window, click **Magnetic Fields (mf)**.

3 In the **Settings** window for Magnetic Fields, click to expand the **Discretization** section.

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

Apply **Ampère's Law** in the core and the air domain.

Ampère's Law 2

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 In the **Settings** window for Ampère's Law, locate the **Domain Selection** section.

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

4 Locate the **Magnetic Field** section. From the **Constitutive relation** list, choose **HB curve**.

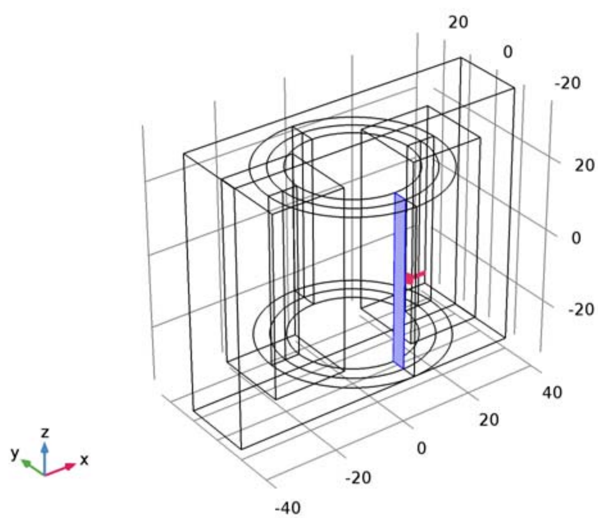
Add **Coil** features to model the primary and the secondary windings using the **Homogenized Multi-Turn** conductor model.

Coil 1

1. On the **Physics** toolbar, click **Domains** and choose **Coil**.
2. In the **Settings** window for Coil, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Primary Winding**.
4. Locate the **Coil** section. From the **Conductor model** list, choose **Homogenized multi-turn**.
5. From the **Coil type** list, choose **Numeric**.
6. Locate the **Homogenized Multi-Turn Conductor** section. In the N text field, type `Np`.
7. Locate the **Coil** section. From the **Coil excitation** list, choose **Circuit (current)**.
8. In the **Model Builder** window, expand the **Coil 1** node.

Input 1

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Magnetic Fields (mf)>Coil 1>Geometry Analysis 1** node, then click **Input 1**.
2. Select Boundary 35 only.



Coil 2

1. On the **Physics** toolbar, click **Domains** and choose **Coil**.

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

3 From the **Selection** list, choose **Secondary Winding**.

4 Locate the **Coil** section. From the **Conductor model** list, choose **Homogenized multi-turn**.

5 From the **Coil type** list, choose **Numeric**.

6 Locate the **Homogenized Multi-Turn Conductor** section. In the N text field, type `Ns`.

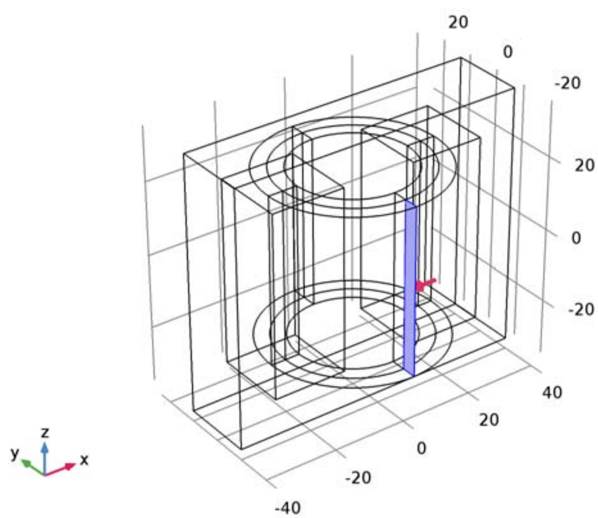
7 Locate the **Coil** section. From the **Coil excitation** list, choose **Circuit (current)**.

8 In the **Model Builder** window, expand the **Coil 2** node.

Input 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Magnetic Fields (mf)>Coil 2>Geometry Analysis 1** node, then click **Input 1**.

2 Select Boundary 31 only.



Gauge Fixing for A-Field 1

1 On the **Physics** toolbar, click **Domains** and choose **Gauge Fixing for A-Field**.

Gauge fixing must be applied on all domains in which the **Magnetic Fields** interface is active.

2 In the **Settings** window for Gauge Fixing for A-Field, locate the **Domain Selection** section.

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

ELECTRICAL CIRCUIT (CIR)

Add the external circuit elements to the primary and the secondary side of the transformer as shown in Figure 2.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electrical Circuit (cir)**.

Voltage Source 1

1 On the **Electrical Circuit** toolbar, click **Voltage Source**.

2 In the **Settings** window for Voltage Source, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	0

4 Locate the **Device Parameters** section. From the **Source type** list, choose **Sine source**.

5 In the V_{src} text field, type `Vac`.

6 In the f text field, type `nu`.

External I Vs. U 1

1 On the **Electrical Circuit** toolbar, click **External I Vs. U**.

2 In the **Settings** window for External I Vs. U, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	1

4 Locate the **External Device** section. From the V list, choose **Coil voltage (mf/coil1)**.

Resistor 1

1 On the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for Resistor, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	0
n	2

4 Locate the **Device Parameters** section. In the R text field, type `Rp`.

External I Vs. U 2

1 On the **Electrical Circuit** toolbar, click **External I Vs. U**.

2 In the **Settings** window for External I Vs. U, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	3
n	0

4 Locate the **External Device** section. From the V list, choose **Coil voltage (mf/coil2)**.

Resistor 2

1 On the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for Resistor, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	0
n	3

4 Locate the **Device Parameters** section. In the R text field, type `Rs`.

Assign materials for the model. Begin by specifying air for all domains.

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

MATERIALS

Air (mat1)

Change the conductivity value to 10 S/m. This small value for conductivity helps to improve the stability of the solver.

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.

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

3 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Electrical conductivity	sigma	`10[S/m]`	S/m	Basic

Next, assign the soft iron (without loss) material for the core. Modify the conductivity value to 10 S/m.

ADD MATERIAL

1. Go to the **Add Material** window.
2. In the tree, select **AC/DC>Soft Iron (without losses)**.
3. Click **Add to Component** in the window toolbar.

MATERIALS

Soft Iron (without losses) (mat2)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Soft Iron (without losses) (mat2)**.
2. In the **Settings** window for Material, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Core**.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Electrical conductivity	sigma	`10[S/m]`	S/m	Basic

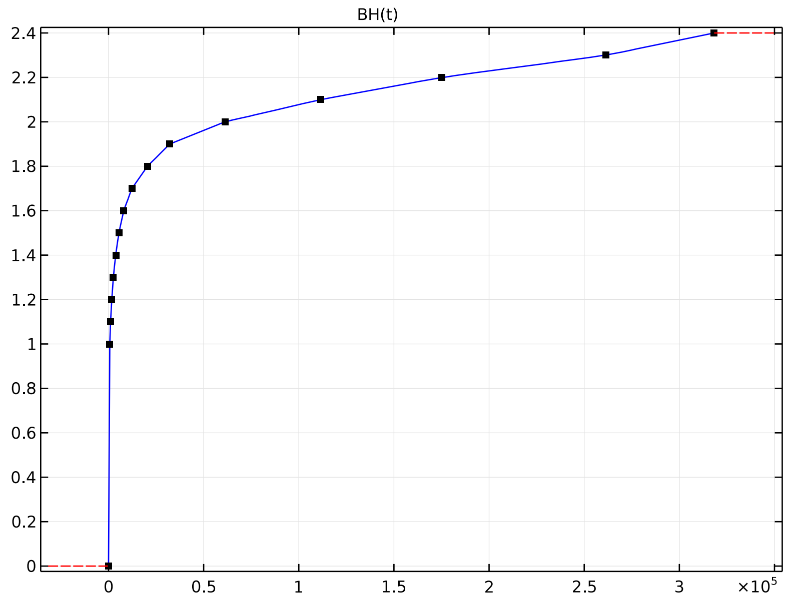
5. On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

   Use the following steps to visualize the nonlinear B-H curve of the soft iron.
6. In the **Model Builder** window, expand the **Soft Iron (without losses) (mat2)** node.

Interpolation (BH)

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Materials>Soft Iron (without losses) (mat2)>BH curve (BHCurve)** node, then click **Interpolation (BH)**.
2. In the **Settings** window for Interpolation, locate the **Interpolation and Extrapolation** section.
3. From the **Extrapolation** list, choose **Constant**.

4 Click **Plot**.



MESH 1

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

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

3 From the **Element size** list, choose **Extra coarse**.

Free Tetrahedral 1

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

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

3 From the **Geometric entity level** list, choose **Entire geometry**.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Tetrahedral 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 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type 2-10 in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for Size, locate the **Element Size** section.

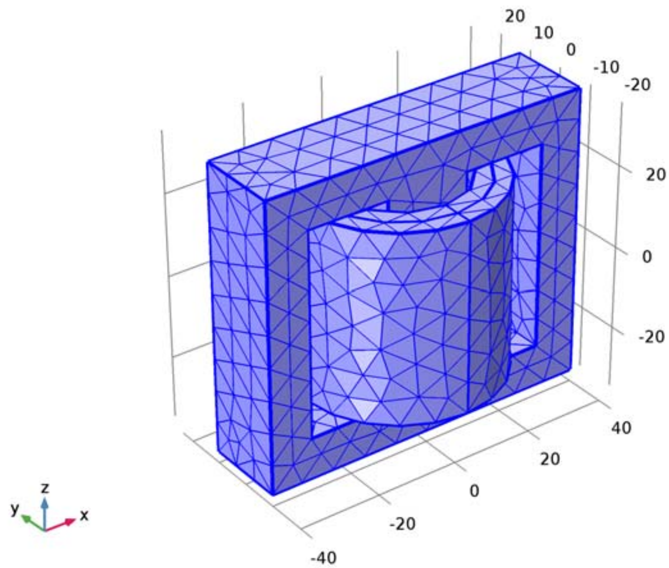
8 Click the **Custom** button.

9 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

10 In the associated text field, type 8.

11 Click **Build All**.

After adjusting the hiding settings by clicking on the **View Unhidden Only** button, the mesh should look like that shown in the figure below. A coarse mesh is used here to solve the model quickly. For real simulations, you are recommended to use a finer mesh.



STUDY 1

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

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

Add the **Coil Geometry Analysis** study step for the analysis of the geometry of the primary and the secondary windings.

Coil Geometry Analysis

On the **Study** toolbar, click **Study Steps** and choose **Other>Coil Geometry Analysis**.

Step 2: Coil Geometry Analysis

In the **Model Builder** window, under **Study 1** right-click **Step 2: Coil Geometry Analysis** and choose **Move Up**.

Step 2: Time Dependent

Solve the problem in time domain from 0 to 50 milliseconds.

1 In the **Model Builder** window, under **Study 1** click **Step 2: Time Dependent**.
2 In the **Settings** window for Time Dependent, locate the **Study Settings** section.
3 In the **Times** text field, type `range(0,5e-4,0.05)`.
4 Select the **Relative tolerance** check box.
5 In the associated text field, type `0.001`.

Follow the steps given below to tune the solver. Such tuning is necessary in order to successfully use a realistic nonlinear B-H curve in a large time-dependent model. As the equations are nonlinear the solver needs to be robust enough to handle such nonlinearities.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** node.
4 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** and choose **Fully Coupled**.
5 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1> Direct** and choose **Enable**.
6 In the **Settings** window for Direct, locate the **General** section.
7 In the **Memory allocation factor** text field, type `2`.
8 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Time-Dependent Solver 1** click **Fully Coupled 1**.
9 In the **Settings** window for Fully Coupled, click to expand the **Method and termination** section.
10 Locate the **Method and Termination** section. From the **Jacobian update** list, choose **On every iteration**.
11 In the **Maximum number of iterations** text field, type `25`.
12 On the **Study** toolbar, click **Compute**.

RESULTS

In the **Model Builder** window, expand the **Results** node.

Study 1/Solution 1 (sol1)

Create separate data sets for the windings and the core domain. This is useful to visualize the results in different domain independently.

1. In the **Model Builder** window, expand the **Results>Data Sets** node.
2. Right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

Selection

1. On the **Results** toolbar, click **Selection**.
2. In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. From the **Selection** list, choose **Primary Winding**.
5. Select the **Propagate to lower dimensions** check box.
6. Right-click **Selection** and choose **Rename**.
7. In the **Rename Selection** dialog box, type `Primary Winding` in the **New label** text field.
8. Click **OK**.

Study 1/Solution 1 (1) (sol1)

In the **Model Builder** window, under **Results>Data Sets** right-click **Study 1/Solution 1 (1) (sol1)** and choose **Duplicate**.

Selection

1. On the **Results** toolbar, click **Selection**.
2. In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. From the **Selection** list, choose **Secondary Winding**.
5. Select the **Propagate to lower dimensions** check box.
6. Right-click **Selection** and choose **Rename**.
7. In the **Rename Selection** dialog box, type `Secondary Winding` in the **New label** text field.
8. Click **OK**.

Study 1/Solution 1 (1) (sol1)

In the **Model Builder** window, under **Results>Data Sets** right-click **Study 1/Solution 1 (1) (sol1)** and choose **Duplicate**.

Selection

1 On the **Results** toolbar, click **Selection**.
2 In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **Core**.
5 Select the **Propagate to lower dimensions** check box.
6 Right-click **Selection** and choose **Rename**.
7 In the **Rename Selection** dialog box, type `Core` in the **New label** text field.
8 Click **OK**.

Selection

1 On the **Results** toolbar, click **Selection**.
2 In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Click **Paste Selection**.
5 In the **Paste Selection** dialog box, type `2,4,6-10` in the **Selection** text field.
6 Click **OK**.

Use the following steps to generate a surface plot of the magnetic flux density norm and arrow plot of the currents in the windings. The figure should look like that shown in Figure 3.

Since the current magnitude in the secondary is much smaller than in the primary, two separate arrow plots (with different scales) will be used.

3D Plot Group 1

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

Arrow Volume 1

1 Right-click **3D Plot Group 1** and choose **Arrow Volume**.
2 In the **Settings** window for Arrow Volume, locate the **Data** section.
3 From the **Data set** list, choose **Study 1/Solution 1 (3) (sol1)**.
4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields>Currents and charge>mf.Jx,mf.Jy,mf.Jz - Current density**.

5 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type 10.

6 Find the **y grid points** subsection. In the **Points** text field, type 10.

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

8 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

Arrow Volume 2

1 Right-click **Results>3D Plot Group 1>Arrow Volume 1** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Study 1/Solution 1 (4) (sol1)**.

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

5 On the **3D Plot Group 1** toolbar, click **Plot**.

Follow the steps below to reproduce the magnetic flux density norm plot as shown in Figure 4.

3D Plot Group 2

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

2 In the **Settings** window for 3D Plot Group, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Solution 1 (5) (sol1)**.

Slice 1

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

2 In the **Settings** window for Slice, locate the **Plane Data** section.

3 From the **Plane** list, choose **zx-planes**.

4 In the **Planes** text field, type 1.

3D Plot Group 2

In the **Model Builder** window, under **Results** right-click **3D Plot Group 2** and choose **Arrow Volume**.

Arrow Volume 1

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

2 Find the **x grid points** subsection. In the **Points** text field, type 10.

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

4 Find the **z grid points** subsection. In the **Points** text field, type 10.

5 Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.

6 Select the **Scale factor** check box.

7 In the associated text field, type `30`.

8 From the **Color** list, choose **Black**.

9 On the **3D Plot Group 2** toolbar, click **Plot**.

Next, plot the induced voltage in the primary winding.

1D Plot Group 3

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

2 In the **Model Builder** window, right-click **1D Plot Group 3** and choose **Rename**.

3 In the **Rename 1D Plot Group** dialog box, type `Induced voltage in primary` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for 1D Plot Group, locate the **Plot Settings** section.

6 Select the **x-axis label** check box.

7 In the associated text field, type `Time (seconds)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Induced voltage in primary (V)`.

Global 1

1 On the **Induced voltage in primary** toolbar, click **Global**.

2 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Magnetic Fields>Coil parameters>mf.VCoil_1 - Coil voltage**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`mf.VCoil_1`	`V`	`Induced voltage in primary winding`

4 Click to expand the **Legends** section. Clear the **Show legends** check box.

5 On the **Induced voltage in primary** toolbar, click **Plot**.

Compare the figure with that shown in Figure 5.

Plot the induced voltage in the secondary winding. The plot is as shown in Figure 6.

1D Plot Group 4

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

2 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Rename**.

3 In the **Rename 1D Plot Group** dialog box, type `Induced voltage in secondary` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for 1D Plot Group, locate the **Plot Settings** section.

6 Select the **x-axis label** check box.

7 In the associated text field, type `Time (seconds)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Induced voltage in secondary (V)`.

Global 1

1 On the **Induced voltage in secondary** toolbar, click **Global**.

2 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Magnetic Fields>Coil parameters>mf.VCoil_2 - Coil voltage**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`mf.VCoil_2`	`V`	`Induced voltage in secondary winding`

4 Locate the **Legends** section. Clear the **Show legends** check box.

5 On the **Induced voltage in secondary** toolbar, click **Plot**.

Plot the current in the primary winding and compare the plot with Figure 7.

1D Plot Group 5

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

2 In the **Model Builder** window, right-click **1D Plot Group 5** and choose **Rename**.

3 In the **Rename 1D Plot Group** dialog box, type `Current in primary winding` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for 1D Plot Group, locate the **Plot Settings** section.

6 Select the **x-axis label** check box.

7 In the associated text field, type `Time (seconds)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Current in primary winding (A)`.

Global 1

1 On the **Current in primary winding** toolbar, click **Global**.
2 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Magnetic Fields>Coil parameters>mf.ICoil_1 - Coil current**.
3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`mf.ICoil_1`	`A`	`Current in primary winding`

4 Locate the **Legends** section. Clear the **Show legends** check box.
5 On the **Current in primary winding** toolbar, click **Plot**.

Next, plot the current in the secondary winding.

1D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Model Builder** window, right-click **1D Plot Group 6** and choose **Rename**.
3 In the **Rename 1D Plot Group** dialog box, type `Current in secondary winding` in the **New label** text field.
4 Click **OK**.
5 In the **Settings** window for 1D Plot Group, locate the **Plot Settings** section.
6 Select the **x-axis label** check box.
7 In the associated text field, type `Time (seconds)`.
8 Select the **y-axis label** check box.
9 In the associated text field, type `Current in secondary winding (A)`.

Global 1

1 On the **Current in secondary winding** toolbar, click **Global**.
2 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Magnetic Fields>Coil parameters>mf.ICoil_2 - Coil current**.
3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`mf.ICoil_2`	`A`	`Current in secondary winding`

4 Locate the **Legends** section. Clear the **Show legends** check box.

5 On the **Current in secondary winding** toolbar, click **Plot**.

Compare the plot with Figure 8.

Modify the model to simulate a step down transformer with a turn ratio of 1000 and $R_p = R_s$. In addition, change the supply voltage to $V_s = 25\text{kV}$

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, expand the **Global Definitions** node, then click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
Rs	100[ohm]	100 Ω	Secondary side resistance
Np	3e5	3E5	Number of turns in primary winding
Vac	25[kV]	2.5E4 V	Supply voltage

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Time Dependent**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

1 In the **Model Builder** window, click **Study 2**.

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

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

Coil Geometry Analysis

On the **Study** toolbar, click **Study Steps** and choose **Other>Coil Geometry Analysis**.

Step 2: Coil Geometry Analysis

In the **Model Builder** window, under **Study 2** right-click **Step 2: Coil Geometry Analysis** and choose **Move Up**.

Step 2: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 2: Time Dependent**.

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

3 In the **Times** text field, type `range(0,5e-4,0.05)`.

4 Select the **Relative tolerance** check box.

5 In the associated text field, type `0.001`.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 3 (sol3)>Time-Dependent Solver 1** node.

4 Right-click **Study 2>Solver Configurations>Solution 3 (sol3)>Time-Dependent Solver 1** and choose **Fully Coupled**.

5 Right-click **Study 2>Solver Configurations>Solution 3 (sol3)>Time-Dependent Solver 1>Direct** and choose **Enable**.

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

7 In the **Memory allocation factor** text field, type `2.1`.

8 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 3 (sol3)>Time-Dependent Solver 1** click **Fully Coupled 1**.

9 In the **Settings** window for Fully Coupled, locate the **Method and Termination** section.

10 From the **Jacobian update** list, choose **On every iteration**.

11 In the **Maximum number of iterations** text field, type `25`.

12 On the **Study** toolbar, click **Compute**.

RESULTS

Plot the induced voltage in the primary winding for a step down transformer.

1D Plot Group 7

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

2 In the **Model Builder** window, right-click **1D Plot Group 7** and choose **Rename**.

3 In the **Rename 1D Plot Group** dialog box, type `Induced voltage in primary-II` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for 1D Plot Group, locate the **Data** section.

6 From the **Data set** list, choose **Study 2/Solution 3 (sol3)**.
7 Locate the **Plot Settings** section. Select the **x-axis label** check box.
8 In the associated text field, type `Time (seconds)`.
9 Select the **y-axis label** check box.
10 In the associated text field, type `Induced voltage in primary (V)`.

Global 1

1 On the **Induced voltage in primary-II** toolbar, click **Global**.
2 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Magnetic Fields>Coil parameters>mf.VCoil_1 - Coil voltage**.
3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`mf.VCoil_1`	`V`	`Induced voltage in primary`

4 Locate the **Legends** section. Clear the **Show legends** check box.
5 On the **Induced voltage in primary-II** toolbar, click **Plot**.
Compare the plot with Figure 9.

Finally, plot the induced voltage in the secondary winding for a step down transformer.

1D Plot Group 8

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Model Builder** window, right-click **1D Plot Group 8** and choose **Rename**.
3 In the **Rename 1D Plot Group** dialog box, type `Induced voltage in secondary-II` in the **New label** text field.
4 Click **OK**.
5 In the **Settings** window for 1D Plot Group, locate the **Data** section.
6 From the **Data set** list, choose **Study 2/Solution 3 (sol3)**.
7 Locate the **Plot Settings** section. Select the **x-axis label** check box.
8 In the associated text field, type `Time (seconds)`.
9 Select the **y-axis label** check box.
10 In the associated text field, type `Induced voltage in secondary (V)`.

Global 1

1 On the **Induced voltage in secondary-II** toolbar, click **Global**.

2 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Magnetic Fields>Coil parameters>mf.VCoil_2 - Coil voltage**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`mf.VCoil_2`	`V`	`Induced voltage in secondary`

4 Locate the **Legends** section. Clear the **Show legends** check box.

5 On the **Induced voltage in secondary-II** toolbar, click **Plot**.

The plot should look like that shown in Figure 10

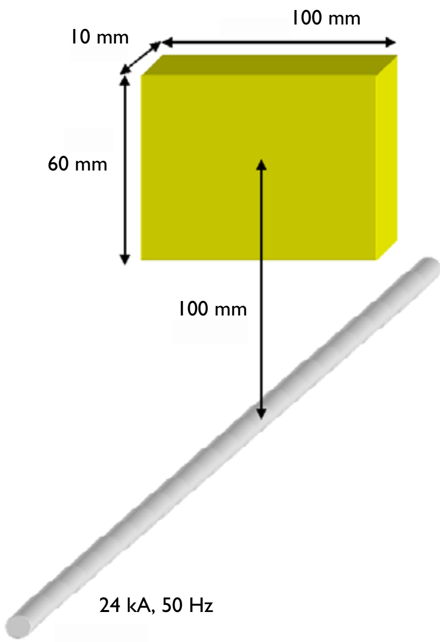
Eddy Currents

Introduction

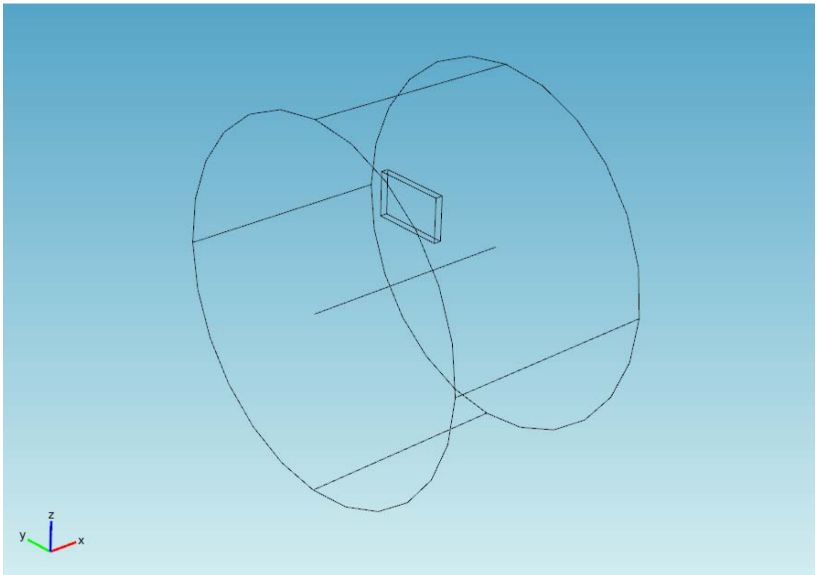
Induced eddy currents and associated thermal loads are of interest in many high power AC applications. This example is of general nature and illustrates some of the involved physics as well as suitable modeling techniques in the AC/DC Module.

Model Definition

A metallic plate is placed near a 50 Hz AC conductor. The resulting eddy current distribution in the plate depends on the conductivity and permeability of the plate. The discussion considers four different materials: copper, aluminum, stainless steel, and magnetic iron. The step-by-step instructions focus on copper and iron. The geometry consists of a single wire and a plate with dimensions as shown below.



Because you cannot afford meshing an infinite volume, it is necessary to specify a finite volume to mesh and solve for. In this case, it makes sense to enclose the wire and the plate in a cylinder with the wire on the axis of this cylinder. This is a good choice when used with the default boundary condition, Magnetic Insulation. This condition forces the field to be tangential to the exterior boundaries, which with the cylindrical shape of the domain is a reasonable approximation to reality.,



The conductor is modeled as a line current with 0° phase and an effective (RMS) value of 24 kA.

The magnetic vector potential is calculated from

$$(j\omega\sigma - \omega^2\varepsilon)\mathbf{A} + \nabla \times \left(\frac{1}{\mu}\nabla \times \mathbf{A}\right) = \mathbf{0}$$

where σ is the conductivity, ε the permittivity, and μ the permeability.

An important parameter in eddy current modeling is the skin depth, δ.

$$\delta = \sqrt{\frac{2}{\omega\mu\sigma}}$$

The following table lists the skin depth for the different materials at a frequency of 50 Hz.

MATERIAL	REL. PERMEABILITY	CONDUCTIVITY	SKIN DEPTH
Copper	1	$5.998\cdot10^7$ S/m	9 mm
Aluminum	1	$3.774\cdot10^7$ S/m	12 mm

MATERIAL	REL. PERMEABILITY	CONDUCTIVITY	SKIN DEPTH
Stainless steel	1	$1.137 \cdot 10^6$ S/m	67 mm
Iron	4000	$1.12 \cdot 10^7$ S/m	0.34 mm

In order for the model to produce accurate results, the mesh needs to resolve the evanescent fields in the metal. In practice, this means you need to resolve the skin depth with at least a bit more than 1 element, preferably closer to 2 or even more. This application uses a maximum element size of 5 mm for the copper plate.

When the skin depth is small in comparison to the size of the conducting objects, it can be practically impossible to resolve the skin depth. This often happens at high frequencies, in large structures, or with highly conductive and permeable materials. These cases require a different technique: Exclude the interior of the conducting objects from the model. Instead, represent them with an impedance boundary condition. This condition essentially sets the skin depth to zero, making all induced currents flow on the surface of the conductors. Mathematically, the relation between the magnetic and electric field at the boundary reads:

$$\mathbf{n} \times \mathbf{H} + \sqrt{\frac{\varepsilon - j\sigma/\omega}{\mu}}\mathbf{n} \times (\mathbf{E} \times \mathbf{n}) = \mathbf{0}$$

The distribution of the dissipated power, P_d (SI unit: W/m^2) can be calculated from

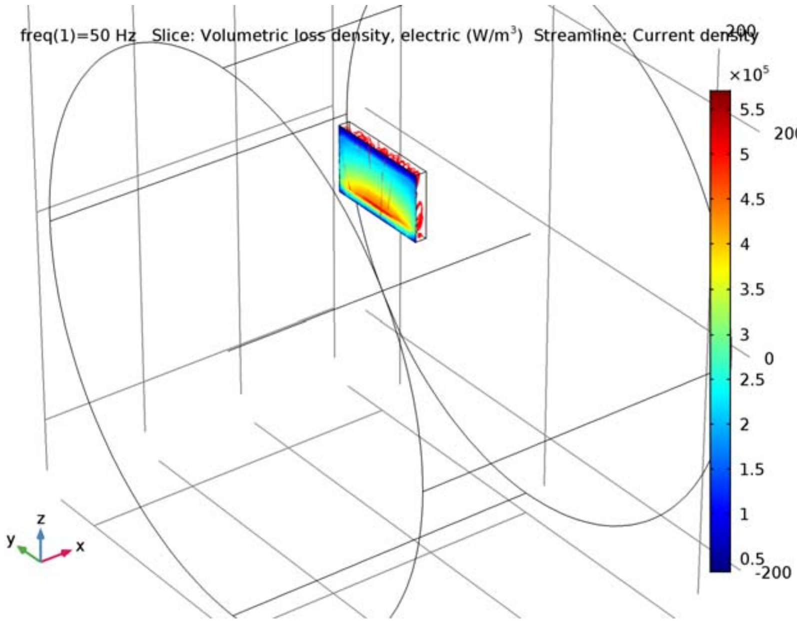
$$P_\mathrm{d} = \frac{1}{2}(\mathbf{J}_S \cdot \mathbf{E}^*)$$

where $\mathbf{J}_S$ is the induced surface current density, and the asterisk (*) denotes the complex conjugate.

This model has the interior of the plate included in the model for copper, and uses the impedance boundary condition for magnetic iron.

Results and Discussion

The induced eddy current distribution for a plate made of copper is shown as streamlines, whereas the distribution of the ohmic losses is shown as a slice plot.



A total dissipated power of 6 W was obtained from integration through the plate. If you repeat the simulation for different materials, the application shows that lowering the conductivity decreases the dissipated power. However, for high permeability materials like soft iron, the dissipated power is higher than in copper (27 W) despite a much lower conductivity.

Application Library path: `ACDC_Module/Inductive_Devices_and_Coils/eddy_currents`

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>Magnetic Fields (mf)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Frequency Domain**.
6 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for Geometry, locate the **Units** section.
3 From the **Length unit** list, choose **mm**.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.
3 In the **Radius** text field, type 250.
4 In the **Height** text field, type 300.
5 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.
6 In the **x** text field, type 1.
7 In the **z** text field, type 0.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Bézier Polygon**.
2 In the **Settings** window for Bézier Polygon, locate the **Polygon Segments** section.
3 Find the **Added segments** subsection. Click **Add Linear**.
4 Find the **Control points** subsection. In row **2**, set **x** to 300.
5 Find the **Added segments** subsection. Click **Add Linear**.
6 Find the **Control points** subsection. Click **Close Curve**.

Block 1 (blk1)

1 On 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 `10`.

4 In the **Depth** text field, type `100`.

5 In the **Height** text field, type `60`.

6 Locate the **Position** section. In the **x** text field, type `145`.

7 In the **y** text field, type `-50`.

8 In the **z** text field, type `70`.

9 Click **Build All Objects**.

The model geometry is now complete.

10 Click the **Transparency** button on the **Graphics** toolbar.

Prepare for your modeling by defining selection groups.

DEFINITIONS

Explicit 1

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

2 In the **Model Builder** window, right-click **Explicit 1** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `Plate` in the **New label** text field.

4 Click **OK**.

5 Select Domain 2 only.

Explicit 2

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

2 In the **Model Builder** window, right-click **Explicit 2** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `Plate Boundaries` in the **New label** text field.

4 Click **OK**.

5 Select Domain 2 only.

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

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

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

MATERIALS

Air (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	`100[S/m]`	S/m	Basic

A small non-zero value of the air conductivity will not substantially affect the results, but helps the solver converge.

Next, override air as the material for the plate domain by copper.

ADD MATERIAL

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

2 In the tree, select **Built-In>Copper**.

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

MATERIALS

Copper (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat2)**.

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

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

MAGNETIC FIELDS (MF)

Edge Current 1

1 On the **Physics** toolbar, click **Edges** and choose **Edge Current**.

2 Select Edge 6 only.

3 In the **Settings** window for Edge Current, locate the **Edge Current** section.

4 In the I_0 text field, type `sqrt(2)*24[kA]`.

This gives you an RMS current of 24 kA.

MESH 1

To resolve the skin depth while maintaining a good mesh economy, mesh the copper plate finer than the rest of the geometry. Note that you need to add the finer mesh first in the sequence; otherwise you would get a coarse mesh on the common domain boundaries that would constrain the mesh inside the copper plate domain.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

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

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

4 From the **Selection** list, choose **Plate**.

5 Click to expand the **Scale geometry** section. Locate the **Scale Geometry** section. In the **y-direction scale** text field, type `0.4`.

6 In the **z-direction scale** text field, type `0.4`.

Size 1

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

2 In the **Settings** window for Size, locate the **Element Size** section.

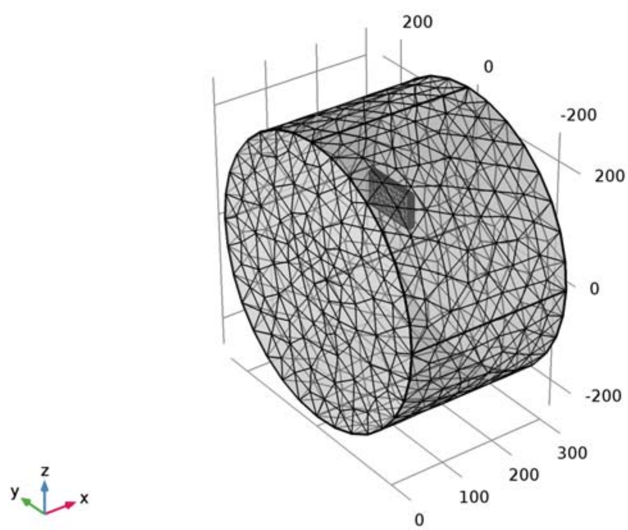
3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

5 In the associated text field, type `5`.

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

7 In the **Settings** window for Mesh, click **Build All**.



STUDY 1

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type 50.

3 On the **Home** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density Norm (mf)

The default plot shows the magnetic flux density norm in three cross sections. The following instructions show you how to visualize the eddy currents and the resistive heating in the copper plate.

Multislice 1

1 In the **Model Builder** window, expand the **Magnetic Flux Density Norm (mf)** node.

2 Right-click **Multislice 1** and choose **Delete**. Click **Yes** to confirm.

Magnetic Flux Density Norm (mf)

In the **Model Builder** window, under **Results** right-click **Magnetic Flux Density Norm (mf)** and choose **Slice**.

Slice 1

1 In the **Settings** window for Slice, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields> Heating and losses>mf.Qrh - Volumetric loss density, electric**.
2 Locate the **Plane Data** section. From the **Entry method** list, choose **Coordinates**.
3 In the **x-coordinates** text field, type `145.1`.
4 On the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.
5 Click the **Transparency** button on the **Graphics** toolbar.

   This returns the transparency setting to its default state.

Magnetic Flux Density Norm (mf)

Right-click **Magnetic Flux Density Norm (mf)** and choose **Streamline**.

Streamline 1

1 In the **Settings** window for Streamline, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields> Currents and charge>mf.Jx,mf.Jy,mf.Jz - Current density**.
2 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Start point controlled**.
3 In the **Points** text field, type `50`.
4 Click to expand the **Advanced** section. In the **Maximum number of integration steps** text field, type `200`.

   This setting reduces the length of the streamlines.
5 On the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

   Due to its nonzero conductivity, the air domain too will contain streamlines. Select to see the streamlines only in the copper plate:

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/Solution 1 (sol1)**.

Selection

1 On the **Results** toolbar, click **Selection**.
2 In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **Plate**.

Magnetic Flux Density Norm (mf)

Zoom in to get a better view of the results.

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

As a final step, integrate the resistive heating in the copper to compute the total heating power.

Volume Integration 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Volume Integration**.

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

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

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Magnetic Fields>Heating and losses>mf.Qrh - Volumetric loss density, electric**.

5 Click **Evaluate**.

The result should be close to 6 W.

If you would like to repeat the analysis for aluminum or some other material with a skin depth of the order of 1 cm or greater, just change the material in the plate and run the simulation again. In the remaining part of this application, you will use the impedance condition to compute the results for magnetic iron, which has a skin depth much less than the thickness of the plate.

MAGNETIC FIELDS (MF)

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

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

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

4 Select Domain 1 only.

Removing the plate means that the equation will not be solved inside it. Add an impedance condition to turn on a surface representation instead.

Impedance Boundary Condition 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Impedance Boundary Condition**.

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

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

ADD MATERIAL

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

2 In the tree, select **Built-In>Iron**.

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

MATERIALS

Iron (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Iron (mat3)**.

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

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

4 From the **Selection** list, choose **Plate Boundaries**.

STUDY 1

To keep the results for copper, disable Solver 1 before computing the study. This way, COMSOL Multiphysics generates a second Solver branch.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

2 Right-click **Solution 1 (sol1)** and choose **Disable**.

3 On the **Home** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density Norm (mf) 1

The resistive heating is now available on the surface of the plate. You can remove the default Multislice plot and add a surface plot of the surface resistive heating.

Multislice 1

1 In the **Model Builder** window, expand the **Magnetic Flux Density Norm (mf) 1** node.

2 Right-click **Multislice 1** and choose **Delete**. Click **Yes** to confirm.

Magnetic Flux Density Norm (mf) 1

In the **Model Builder** window, under **Results** right-click **Magnetic Flux Density Norm (mf) 1** and choose **Surface**.

Surface 1

1 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields>Heating and losses>mf.Qsrh - Surface loss density, electric**.

2 On the **Magnetic Flux Density Norm (mf) 1** toolbar, click **Plot**.

Study 1/Solution 2 (sol2)

In the **Model Builder** window, under **Results>Data Sets** click **Study 1/Solution 2 (sol2)**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 From the **Selection** list, choose **Plate Boundaries**.

Surface Integration 2

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.

2 In the **Settings** window for Surface Integration, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Solution 2 (sol2)**.

4 Locate the **Selection** section. From the **Selection** list, choose **Plate Boundaries**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Magnetic Fields>Heating and losses>mf.Qsrh - Surface loss density, electric**.

6 Click **Evaluate**.

The result should be close to 27 W.

Created in COMSOL Multiphysics 5.2a

Electric Shielding Comparison

Introduction

The electric shielding boundary condition is meant to approximate a thin layer of highly conductive material that provides an additional current path tangential to a boundary. This example compares the electric shielding boundary condition to a full-fidelity model and discusses the range of applicability of this boundary condition.

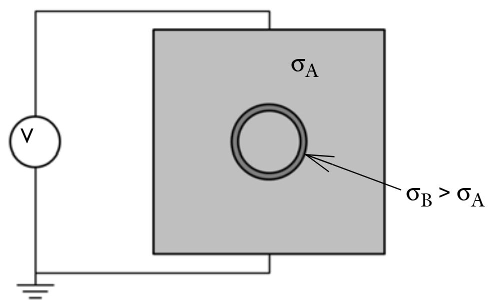


Figure 1: A square 2D domain of conductive material, with a circular inclusion. The wall of the inclusion are made of a material with higher conductivity.

Model Definition

The situation being modeled is shown in Figure 1. A 1 m square two-dimensional domain of conductive material has a DC voltage difference applied to it. Within the square domain, there is a circular inclusion of radius 0.25 m. The 0.01 m thick walls of this inclusion are modeled two ways, first using a full fidelity model that includes the thickness of the walls, and also using the electric shielding boundary condition. The inside of the inclusion has the same properties as the bulk.

The location of the electric shielding condition is at the center line, midway between the inner and outer radii of the full fidelity model. Note that, when using the electric shielding condition, the total volume of the surrounding material is slightly larger, since the thickness of the wall is not being explicitly modeled. The conductivity of the wall of the inclusion is varied.

Results and Discussion

The voltage distribution and the electric field strength is plotted in Figure 2 for the case where the electric conductivity is one thousand times greater in the wall of the inclusion than in the bulk. This represents a thin walled object that allows significant current flow along its surface, that is, it shields whatever is inside the inclusion from the electric fields and current flow. The current can be observed to flow towards the inclusion, and then along the surface. The solutions for the full fidelity and electric shielding model agree well for the cases where the conductivity is greater than the surrounding medium.

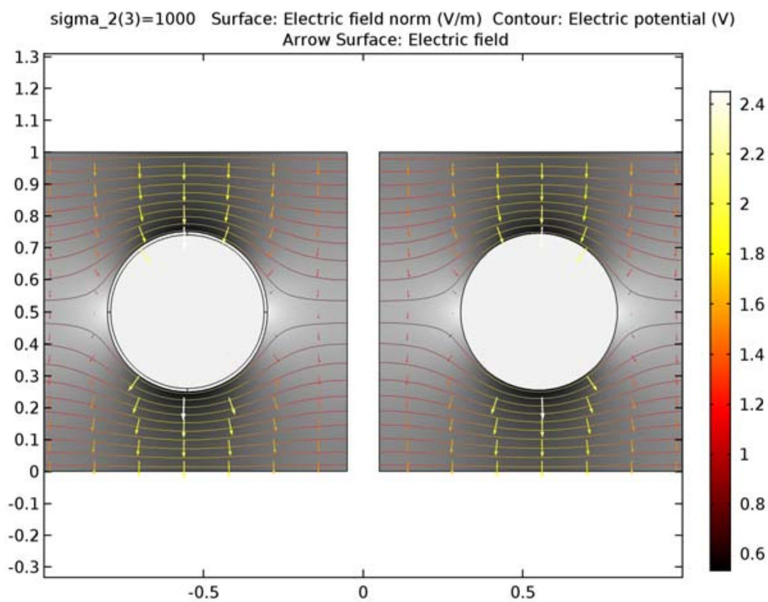


Figure 2: Isolines of the voltage field, arrow plot of the current flow, and a grayscale plot of the electric field for the case of a thin layer of material that has high conductivity. The full fidelity (left) and electric shielding (right) solutions are almost identical.

Figure 3 shows the case where the electric conductivity in the wall of the inclusion is only ten times greater than the surroundings. The electric shielding condition still agrees well with the full fidelity model.

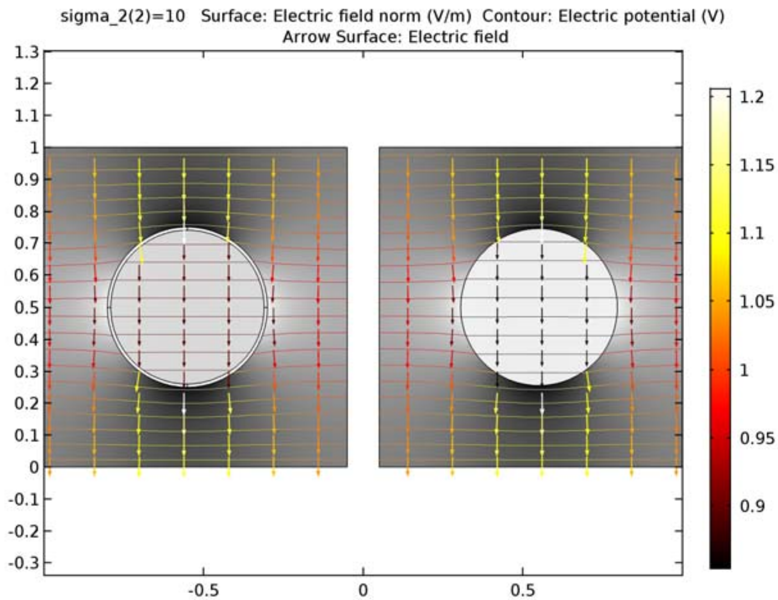


Figure 3: For the case of a thin layer of material that has conductivity ten times greater than the surrounding material.

Figure 4 shows the case where the electric conductivity in the wall of the inclusion is ten times less than the surroundings. The electric shielding condition no longer agrees with the full fidelity model. Although an additional conduction path has been added tangential to the boundary, the magnitude of this conduction path is less than in the surrounding material, thus it does not noticeably alter the current flow. The electric shielding boundary condition also does not impede the current flow normal to the boundary, which is the case, as can be seen in the full fidelity model. The Contact Impedance boundary condition would be more appropriate for this case.

The electric shielding boundary condition can be used in cases where the thickness of the boundary being approximated is much smaller than the characteristic size of the model domain, and when the conductivity of the layer is greater than the surrounding medium.

When this boundary condition can be used, the resulting mesh size is much smaller, saving solution time and memory.

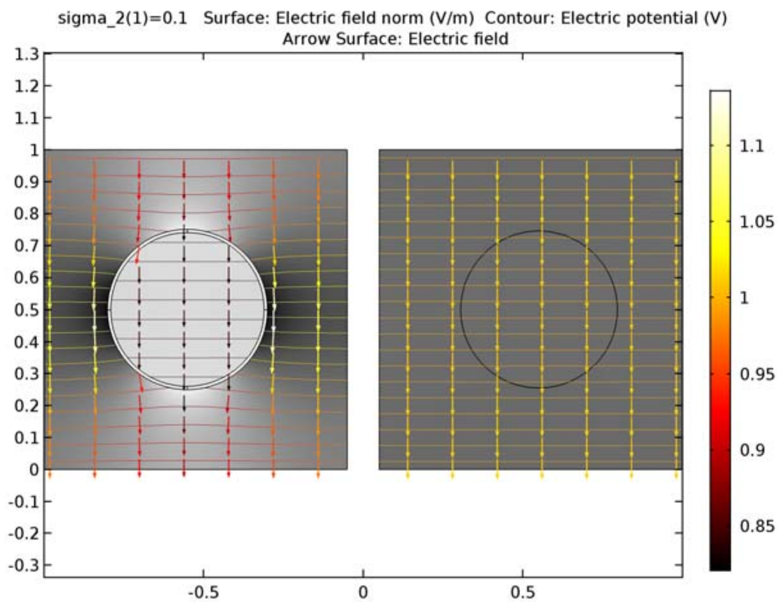


Figure 4: For the case of a thin layer of material that has conductivity ten times less than the surrounding material. The solutions do not agree for this case.

Application Library path: `ACDC_Module/Resistive_Devices/electric_shielding_comparison`

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 **2D**.

2 In the **Select Physics** tree, select **AC/DC>Electric Currents (ec)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`sigma_1`	`1[S/m]`	1 S/m	`Conductivity, material 1`
`sigma_2`	`1[S/m]`	1 S/m	`Conductivity, material 2`

GEOMETRY 1

Square 1 (sq1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Square**.

2 In the **Settings** window for Square, locate the **Position** section.

3 In the **x** text field, type `0.05`.

Square 2 (sq2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Square**.

2 In the **Settings** window for Square, locate the **Position** section.

3 In the **x** text field, type `-1.05`.

4 Click **Build All Objects**.

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

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `0.245`.

4 Locate the **Position** section. In the **x** text field, type `0.55`.

5 In the **y** text field, type `0.5`.

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `0.25`.

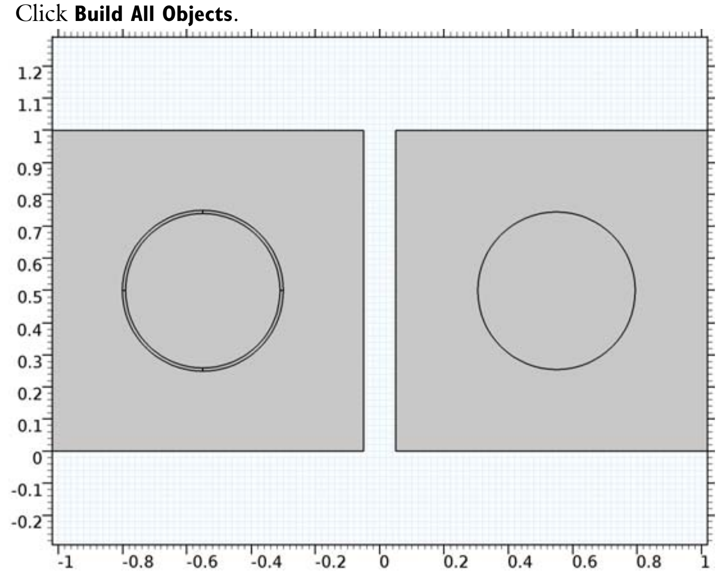
4 Locate the **Position** section. In the **x** text field, type `-0.55`.

5 In the **y** text field, type `0.5`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`0.01`

7 Click **Build All Objects**.



The geometry on the left side describes the full fidelity model. The geometry on the right side replaces the thin layer with a boundary in order to use the **Electric Shielding** feature.

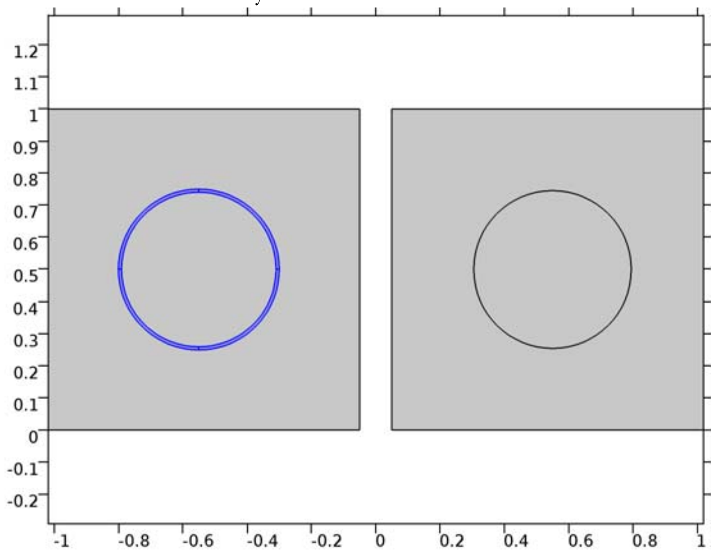
DEFINITIONS

Create a set of selections for use before setting up the physics. First, create a selection for the wall of the inclusion.

Explicit 1

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

2 Select Domains 2–5 only.



3 Right-click **Explicit 1** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Full fidelity` in the **New label** text field.

5 Click **OK**.

Add a selection for the electric shielding boundaries.

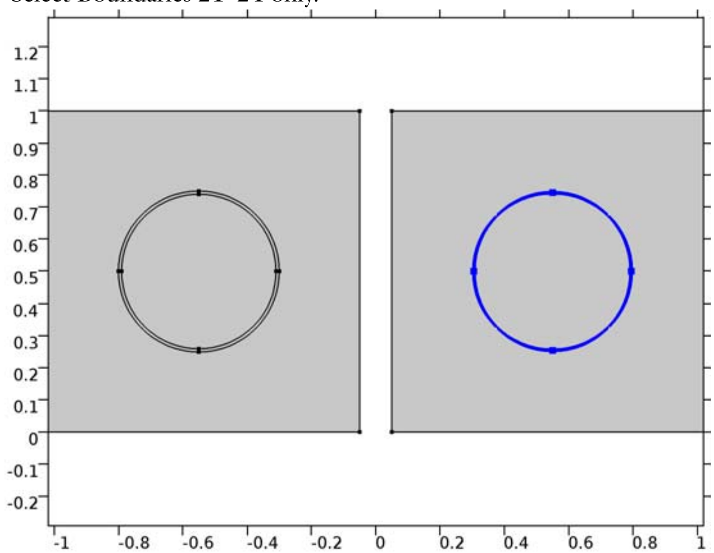
Explicit 2

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

2 In the **Settings** window for Explicit, locate the **Input Entities** section.

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

4 Select Boundaries 21–24 only.



5 Right-click **Explicit 2** and choose **Rename**.

6 In the **Rename Explicit** dialog box, type `Electric shielding` in the **New label** text field.

7 Click **OK**.

Add a selection for the bulk area. This is the complementary domain of the wall of the inclusion.

Complement 1

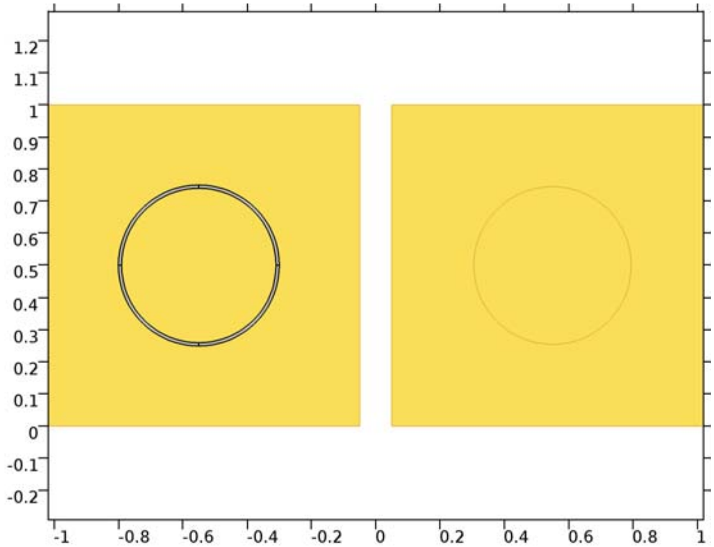
1 On the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for Complement, locate the **Input Entities** section.

3 Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **Full fidelity** in the **Selections to invert** list.

5 Click **OK**.



6 Right-click **Complement 1** and choose **Rename**.

7 In the **Rename Complement** dialog box, type `Bulk` in the **New label** text field.

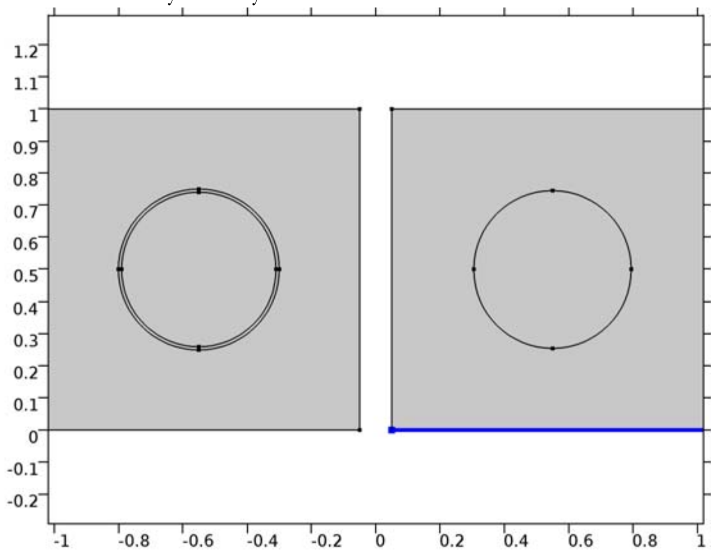
8 Click **OK**.

ELECTRIC CURRENTS (EC)

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

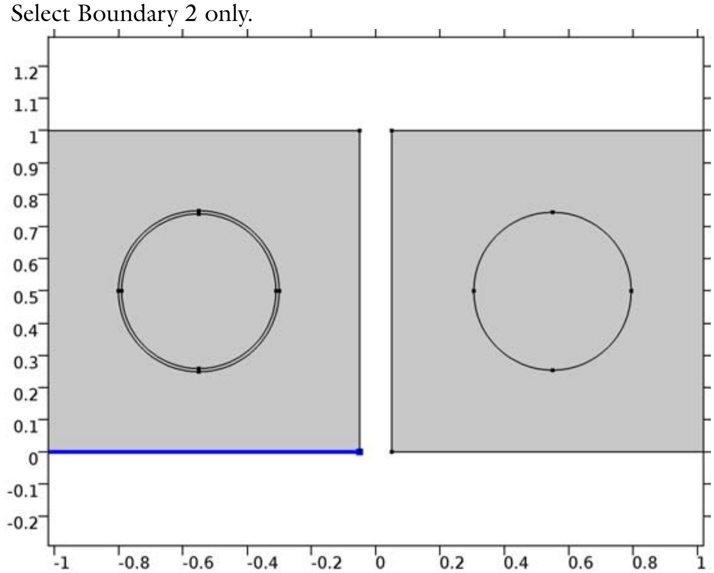
2 Select Boundary 10 only.



Ground 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

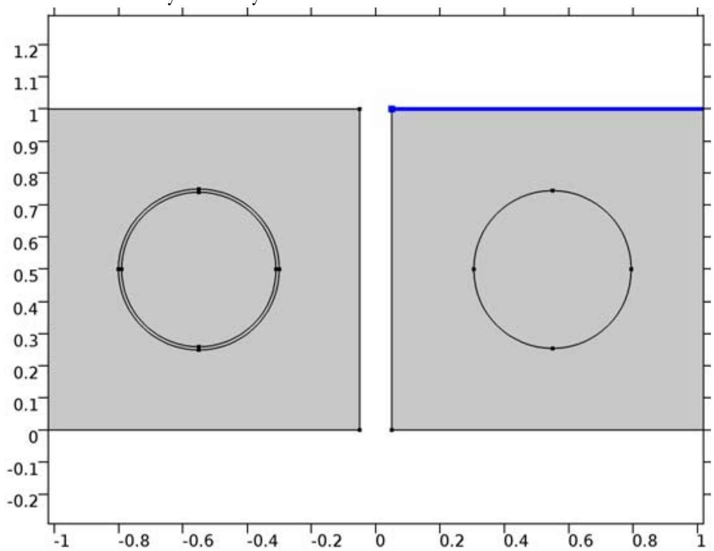
2 Select Boundary 2 only.



Terminal 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Terminal**.

2 Select Boundary 11 only.



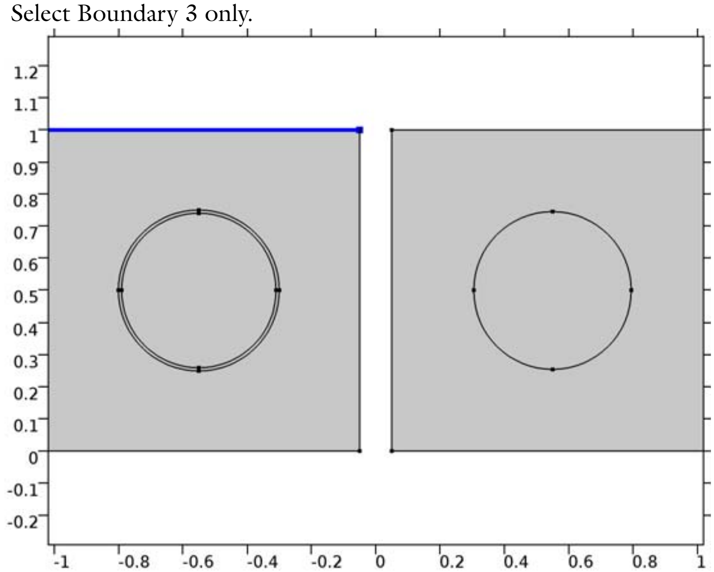
3 In the **Settings** window for Terminal, locate the **Terminal** section.

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

Terminal 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Terminal**.

2 Select Boundary 3 only.



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

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

Electric Shielding 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Shielding**.

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

3 From the **Selection** list, choose **Electric shielding**.

4 Locate the **Thickness** section. In the d_s text field, type `1[cm]`.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

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

2 From the **Selection** list, choose **Bulk**.

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

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	`sigma_1`	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic

Material 2 (mat2)

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

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

3 From the **Selection** list, choose **Full fidelity**.

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

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	`sigma_2`	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic

Material 3 (mat3)

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

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

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

4 From the **Selection** list, choose **Electric shielding**.

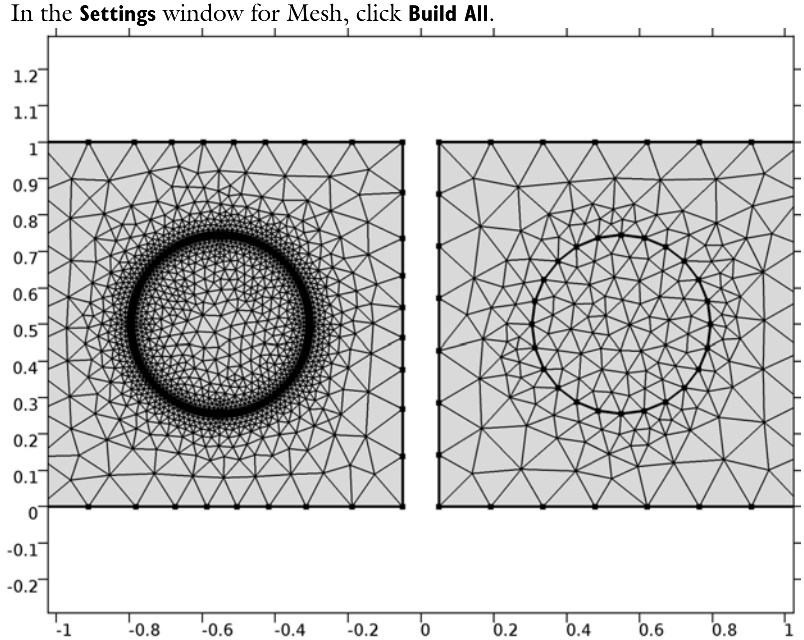
5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	1	1	Basic
Electrical conductivity	sigma	sigma_2	S/m	Basic

MESH 1

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

2 In the **Settings** window for Mesh, click **Build All**.



STUDY 1

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
sigma_2	1e-1 1e1 1e3	

5 On the **Study** toolbar, click **Compute**.

RESULTS

Study 1/Solution 1 (sol1)

Begin the result analysis by suppressing the domain of the wall of the inclusion which is not of interest.

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Solution 1 (sol1)**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 From the **Selection** list, choose **Bulk**.

The default plot shows the surface plot of the electric potential. Change the expression to show the norm of the electric field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Electric Potential (ec)** node, then click **Surface 1**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Electric Currents> Electric>ec.normE - Electric field norm**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayPrint**.

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

5 Select the **Reverse color table** check box.

Electric Potential (ec)

Next, add a contour plot showing the electric potential.

1 In the **Model Builder** window, under **Results** right-click **Electric Potential (ec)** and choose **Contour**.

Contour 1

In the **Model Builder** window, under **Results>Electric Potential (ec)** right-click **Contour 1** and choose **Color Expression**.

Color Expression 1

1 In the **Settings** window for Color Expression, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Electric Currents>Currents and charge>ec.normJ - Current density norm**.

2 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalEquidistant**.

Electric Potential (ec)

Then, add an arrow plot of the electric field.

Arrow Surface 1

1 In the **Model Builder** window, under **Results** right-click **Electric Potential (ec)** 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 **Model>Component 1>Electric Currents>Electric>ec.Ex,ec.Ey - Electric field**.

Color Expression 1

1 Right-click **Results>Electric Potential (ec)>Arrow Surface 1** and choose **Color Expression**.

2 In the **Settings** window for Color Expression, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Electric Currents>Currents and charge>ec.normJ - Current density norm**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalEquidistant**.

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

5 On the **Electric Potential (ec)** toolbar, click **Plot**.

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

Compare the plot with Figure 2.

Electric Potential (ec)

1 In the **Model Builder** window, under **Results** click **Electric Potential (ec)**.

2 In the **Settings** window for 2D Plot Group, locate the **Data** section.

3 From the **Parameter value (sigma_2)** list, choose **10**.

4 On the **Electric Potential (ec)** toolbar, click **Plot**.

This should look like Figure 3.

Finish by reproducing Figure 4.

5 From the **Parameter value (sigma_2)** list, choose **0.1**.

6 On the **Electric Potential (ec)** toolbar, click **Plot**.

Created in COMSOL Multiphysics 5.2a

Electron Beam Divergence Due to Self Potential

Introduction

When modeling the propagation of a charged particle beam at a high current, the electric field due to the space charge of the beam significantly affects the trajectories of the charged particles. Perturbations to these trajectories, in turn, affect the space charge distribution. In order to accurately predict the properties of the beam, the particle trajectories and fields must be computed in a self-consistent manner. The Charged Particle Tracing interface can use an iterative procedure to efficiently compute the strongly coupled particle trajectories and electric field for systems operating under steady-state conditions. Such a procedure reduces the required number of model particles by several orders of magnitude, compared to methods based on explicit modeling of Coulomb interactions between the beam particles. A mesh refinement study confirms that the solution agrees with the analytical expression for the shape of a nonrelativistic, paraxial beam envelope.

Note: This application requires the Particle Tracing Module.

Model Definition

This model computes the shape of an electron beam propagating through free space. When the magnitude of the beam current is large enough that Coulomb interactions are significant, the shape of the beam may be determined by solving a set of strongly coupled equations for the beam potential and the electron trajectories,

$$-\nabla \cdot \varepsilon_0 \nabla V = \sum_{i=1}^{N} eZ\delta(\mathbf{r} - \mathbf{q}_i)$$

$$\frac{d}{dt}(mv) = -eZ\nabla V$$

The beam electrons are assumed to be nonrelativistic so that magnetic forces can be neglected. Modeling the beam electrons and the resulting electric potential using a time-dependent study would require a very large number of model particles to be released at a large number of time intervals. Instead, this model computes the shape of the electron beam by coupling a time-dependent study step for computing the particle trajectories to a stationary step for computing the electric potential. This algorithm is suitable for modeling beams which operate at steady-state conditions. It consists of the following steps:

1 Compute the particle trajectories, assuming no space charge effects are present, using a time-dependent solver. From these trajectories, compute the space charge density using the **Electric Particle Field Interaction** node.

2 Compute the electric potential due to the space charge density of the beam, using a stationary solver. The model uses an **Infinite Element Domain** region to apply appropriate boundary conditions for a beam propagating in free space.

3 Use the electric potential calculated in step 2 to compute the perturbed particle trajectories. Recalculate the space charge density using these perturbed trajectories.

4 Repeat steps 2 and 3 until a specified number of iterations has been reached.

After several iterations, the particle trajectories and the corresponding space charge density and electric field reach a stable, self-consistent solution. For a nonrelativistic, paraxial beam of electrons, the shape of the beam envelope is given by Ref. 1 as

$$z = \frac{R_0 F(\chi)}{\sqrt{2K}} \tag{1}$$

where z is the distance from the beam waist, R_0 is the waist radius, K is the generalized beam perveance,

$$K = \frac{eI_0}{2\pi\varepsilon_0 m_e v_z^3}$$

χ is the ratio of the beam radius to the beam waist radius, and

$$F(\chi) = \int_1^{\chi} \frac{dy}{\sqrt{\ln(y)}} \tag{2}$$

This analytical expression for the relationship between axial position and beam envelope radius is used to determine the accuracy of the solution. A mesh refinement study confirms that the agreement between the solutions improves as the mesh element size is reduced.

In this example, **Specify current** is selected from the **Particle release specification** list in the physics interface **Particle Properties** section. As a result, each model particle represents a continuous stream of particles, released at regular time intervals, rather than the instantaneous position of a charge. For the purpose of modeling particle-field interactions, each model particle leaves behind a trail of space charge in its wake. The contribution of each model particle to the total space charge density of the beam is found by evaluating the sum

$$\frac{d\rho}{dt} = eZ\sum_{i=1}^{N} f_{\mathrm{rel}} d(r - q_i)$$

where e is the elementary charge, Z is the charge number of the particles, δ is the Dirac delta function, and f_{rel} is the effective frequency of release of the particle. The frequency of release is the number of particles per model particle per second. To avoid the infinite potential associated with an infinitesimally small point charge, the space charge density is distributed uniformly over each mesh element before the electrostatics problem is solved.

Results and Discussion

After several iterations, the model reaches a self-consistent solution for the trajectories of the electrons and the beam potential. The electron trajectories are plotted in Figure 1. The expression `r-at(0,r)` is used to define a color expression for the trajectories. The `at()` operator is used to evaluate an expression at the initial time, rather than the current time. Thus the color expression gives the radial displacement of each particle due to space charge effects.

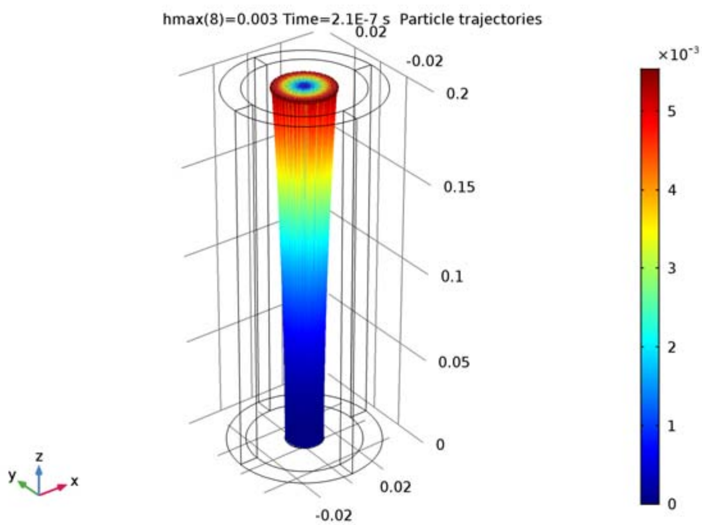


Figure 1: A beam of electrons with a waist located at z = 0 diverges due to transverse beam forces. The color represents the radial displacement of each electron from its initial position.

The electric potential distribution in the beam is shown in Figure 2. Since the beam propagates from left to right, and the beam electrons initially move in the positive

z direction, the left end of the plot corresponds to the beam waist. This is also the location where the beam radius is greatest in magnitude.

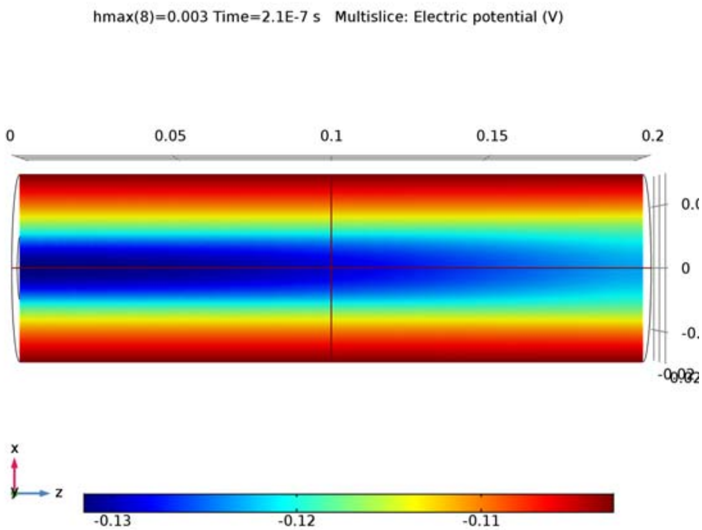


Figure 2: Plot of the electric potential of the electron beam. The potential is greatest in magnitude close to the beam waist.

A mesh refinement study confirms that the shape of the beam envelope agrees more closely with the analytical expression of Equation 1 as the maximum mesh element size is reduced. The distance from the beam waist as a function of beam radius is compared to the result of this expression, and the relative error is plotted in Figure 3. For all values of the maximum element size, the error shown is computed after three iterations of the solver loop.

These results show that a self-consistent solution for the particle trajectories and the fields due to their space charge density can be obtained using an iterative solver sequence. This requires much less time and memory than a fully coupled time-dependent study of the individual beam particles and their fields. The accuracy of the solution clearly improves as the mesh element size is reduced, enabling more accurate computation of the electron trajectories and beam potential.

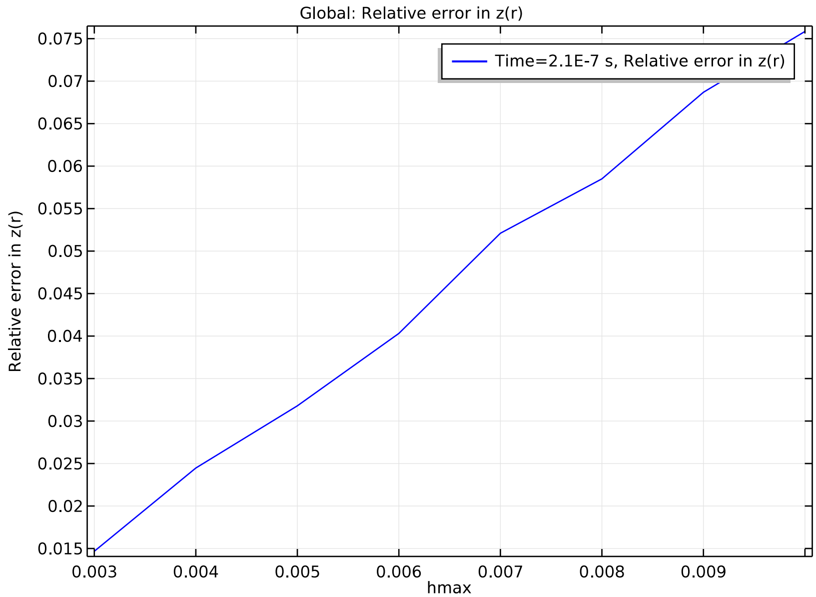


Figure 3: The results of a mesh refinement study indicate that the observed relationship between beam radius and distance from the beam waist converges to the expected value as the mesh is refined. The comparison is made at a location 0.2 meters from the waist.

Reference

1. S. Humphries, *Charged Particle Beams*, Dover Publications, New York, 2013.

Application Library path: `ACDC_Module/Particle_Tracing/electron_beam_divergence`

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>Particle Tracing>Particle Field Interaction, Non-Relativistic**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Bidirectionally Coupled Particle Tracing**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

  To save time, the parameters can be loaded from a file.

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

3 Click **Load from File**.

4 Browse to the application's Application Libraries folder and double-click the file `electron_beam_divergence_parameters.txt`.

GEOMETRY 1

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r2`.

4 In the **Height** text field, type `L`.

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	**Thickness (m)**
`Layer 1`	`tlayer`

Work Plane 1 (wp1)

1 Right-click **Cylinder 1 (cyl1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Work Plane**.

3 In the **Settings** window for Work Plane, locate the **Plane Definition** section.

4 From the **Plane type** list, choose **Face parallel**.

5 Find the **Planar face** subsection. Select the **Active** toggle button.

6 On the object **cyl1**, select Boundary 10 only.

7 Click **Show Work Plane**.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `r0beam`.

4 Right-click **Circle 1 (c1)** and choose **Build Selected**.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

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

2 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	1	1	Basic

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
qr	sqrt(qx^2+qy^2)	m	Radial distance from beam axis
qrmax	cpt.cptmaxop1(qr)	m	Beam radius
z_avg	cpt.cptaveop1(qz)	m	Average z-coordinate
chi	qrmax/at(0,qrmax)		Ratio of beam radius to waist radius

To model a drifting electron beam propagating in a large, open area, surround the modeling domain with an **Infinite Element Domain**. This results in appropriate boundary conditions for the electric potential.

Infinite Element Domain 1 (ie1)

1. On the **Definitions** toolbar, click **Infinite Element Domain**.
2. In the **Settings** window for Infinite Element Domain, locate the **Geometry** section.
3. From the **Type** list, choose **Cylindrical**.
4. Select Domains 1, 2, 4, and 5 only.

CHARGED PARTICLE TRACING (CPT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Charged Particle Tracing (cpt)**.

Inlet 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. Select Boundary 12 only.
3. In the **Settings** window for Inlet, locate the **Release Current Magnitude** section.
4. In the I text field, type `Ibeam`.
5. Locate the **Initial Position** section. From the **Initial position** list, choose **Density**.
6. In the N text field, type `1000`.
7. Locate the **Initial Velocity** section. Specify the $\mathbf{v}_0$ vector as

0	x
0	y
v0beam	z

Electric Force 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Charged Particle Tracing (cpt)** click **Electric Force 1**.
2. Select Domain 3 only.
3. In the **Settings** window for Electric Force, locate the **Electric Force** section.
4. From the $\mathbf{E}$ list, choose **Electric field (es/ccn1)**.
5. Locate the **Advanced Settings** section. Select the **Use piecewise polynomial recovery on field** check box.

ELECTROSTATICS (ES)

On the **Physics** toolbar, click **Charged Particle Tracing (cpt)** and choose **Electrostatics (es)**.

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

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.
2 Select Boundaries 2, 3, 14, and 22 only.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

Free Tetrahedral 1

1 In the **Settings** window for Free Tetrahedral, locate the **Domain Selection** section.
2 From the **Geometric entity level** list, choose **Domain**.
3 Select Domain 3 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Tetrahedral 1** and choose **Size**.
2 In the **Settings** window for Size, locate the **Element Size** section.
3 Click the **Custom** button.
4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
5 In the associated text field, type `hmax`.
6 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Swept 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Swept 1** and choose **Distribution**.

Distribution 1

In the **Settings** window for Distribution, click **Build All**.

STUDY 1

Step 1: Bidirectionally Coupled Particle Tracing

1 In the **Settings** window for Bidirectionally Coupled Particle Tracing, locate the **Study Settings** section.
2 Click **Range**.
3 In the **Range** dialog box, type `1e-8` in the **Step** text field.

4 In the **Stop** text field, type `21e-8`.
5 Click **Replace**.
6 In the **Settings** window for Bidirectionally Coupled Particle Tracing, locate the **Iterations** section.
7 In the **Number of iterations** text field, type `3`.

Use a **Parametric Sweep** to confirm that a finer mesh results in closer agreement with the expected beam envelope size.

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.
2 In the **Settings** window for Parametric Sweep, locate the **Study Settings** section.
3 Click **Add**.
4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
hmax		

5 Click **Range**.
6 In the **Range** dialog box, type `0.01` in the **Start** text field.
7 In the **Step** text field, type `-0.001`.
8 In the **Step** text field, type `0.003`.
9 Click **Replace**.

The smallest mesh size requires about 5 GB RAM.

10 On the **Study** toolbar, click **Compute**.

RESULTS

Plot the trajectories of the electrons, using a **Color Expression** to observe their radial displacement over time.

Particle Trajectories 1

1 In the **Model Builder** window, expand the **Results>Particle Trajectories (cpt)** node, then click **Particle Trajectories 1**.
2 In the **Settings** window for Particle Trajectories, locate the **Coloring and Style** section.
3 Find the **Line style** subsection. From the **Type** list, choose **Line**.

Color Expression 1

1. In the **Model Builder** window, expand the **Particle Trajectories 1** node, then click **Color Expression 1**.
2. In the **Settings** window for Color Expression, locate the **Expression** section.
3. In the **Expression** text field, type `qr-at(0,qr)`.
4. On the **Particle Trajectories (cpt)** toolbar, click **Plot**.
5. Click the **Zoom Extents** button on the **Graphics** toolbar. This plot should look like Figure 1.

Data Sets

The electric potential can be visualized more clearly by excluding the selection of the **Infinite Element Domain** from the **Study 1/Parametric Solutions 1 (sol2)** data set.

Study 1/Parametric Solutions 1 (sol2)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Parametric Solutions 1 (sol2)**.

Selection

1. On the **Results** toolbar, click **Selection**.
2. In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domain 3 only.

Electric Potential (es)

1. In the **Model Builder** window, under **Results** click **Electric Potential (es)**.
2. In the **Settings** window for 3D Plot Group, click to expand the **Color legend** section.
3. Locate the **Color Legend** section. From the **Position** list, choose **Bottom**.
4. Click the **Go to ZX View** button on the **Graphics** toolbar. This plot should look like Figure 2.

For each mesh size, compare the beam envelope shape to the analytical solution for a paraxial, nonrelativistic beam.

1D Plot Group 3

1. On the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for 1D Plot Group, locate the **Data** section.
3. From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.
4. From the **Time selection** list, choose **Last**.

Global 1

1. On the **1D Plot Group 3** toolbar, click **Global**.
2. In the **Settings** window for Global, type `Relative Error in Longitudinal Beam Displacement` in the **Label** text field.
3. Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`abs((r0beam/sqrt(2*K)*integrate(1/sqrt(log(s)),s,1+eps,chi)-z_avg)/z_avg)`		`Relative error in z(r)`

   This expression is Equation 1, where the first argument of the `integrate` operator is the integrand of Equation 2. The lower limit of integration is increased by the floating point relative accuracy `eps` (machine epsilon, 2^{-52} or about 2.2204×10^{-16}) to avoid division by zero during numerical integration.

4. Locate the **x-Axis Data** section. From the **Axis source data** list, choose **hmax**.
5. On the **1D Plot Group 3** toolbar, click **Plot**. This plot should look like Figure 3.

Created in COMSOL Multiphysics 5.2a

Relativistic Paraxial Electron Beam

Introduction

When modeling the propagation of charged particle beams at high currents and relativistic speeds, the space charge and beam current create significant electric and magnetic forces that tend to expand and focus the beam, respectively. The Charged Particle Tracing interface can use an iterative procedure to efficiently compute the strongly coupled particle trajectories and electric and magnetic fields for a beam operating at constant current. A mesh refinement study confirms that the solution agrees with the analytical expression for the shape of a relativistic beam envelope.

Note: This application requires the AC/DC Module and the Particle Tracing Module.

Model Definition

This model is almost identical to the Electron Beam Divergence Due to Self Potential model but with higher beam current and particle velocities. To accurately compute the trajectories of relativistic particles, a correction has to be applied on the mass of the electrons,

$$m_e = \frac{m_r}{\sqrt{1-\frac{v^2}{c^2}}} \tag{1}$$

Where m_r is the electron rest mass, v is the electron speed, and c is the speed of light in vacuum.

At relativistic speeds, the electron beam generates a magnetic field that exerts a significant magnetic force on the electrons. The ratio of self-induced magnetic and electric forces is proportional to $\beta^2 = (v/c)^2$ (Ref. 1).

As for the non-relativistic case, the shape of the beam envelope can be obtained using

$$z = \frac{R_0 F(\chi)}{\sqrt{2K}} \tag{2}$$

where z is the distance from the beam waist, R_0 is the waist radius, and K is the generalized beam perveance:

$$K = \frac{eI_0}{2\pi\varepsilon_0 m_e (v_z\gamma)^3}$$

where γ is the relativistic factor defined as

$$\gamma = \frac{1}{\sqrt{1 - \frac{v^2}{c^2}}} \tag{3}$$

χ is the ratio of the beam radius to the beam waist radius, and

$$F(\chi) = \int_1^{\chi} \frac{dy}{\sqrt{\ln(y)}} \tag{4}$$

This analytical expression for the relationship between axial position and beam envelope radius is used to determine the accuracy of the solution.

Results and Discussion

The electron trajectories are plotted in Figure 1 while the electric potential distribution and magnetic flux norm are respectively shown on Figure 2, and Figure 3. The distance from the beam waist as a function of beam radius is compared to the result of Equation 2, and the relative error is plotted in Figure 4 for all values of the maximum element size. The error shown is computed after three iterations of the solver loop. The accuracy of the solution improves as the mesh element size is reduced, enabling more accurate computation of the electron trajectories and beam potential.

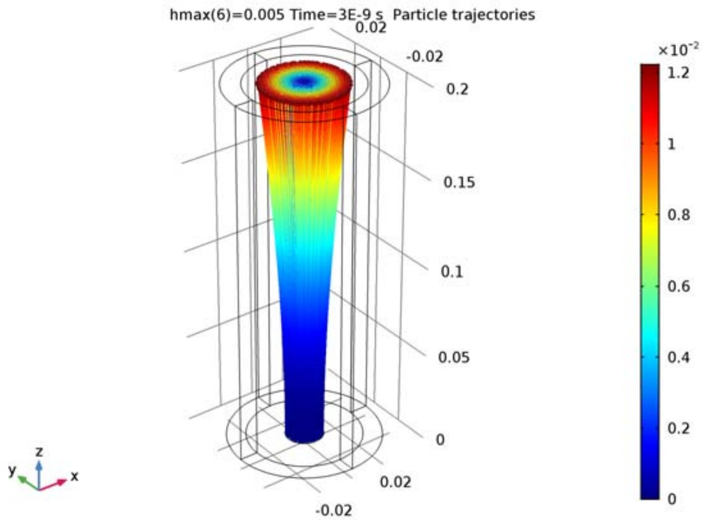


Figure 1: A beam of electrons with a waist located at z = 0 diverges due to transverse beam forces. The color represents the radial displacement of each electron from its initial position.

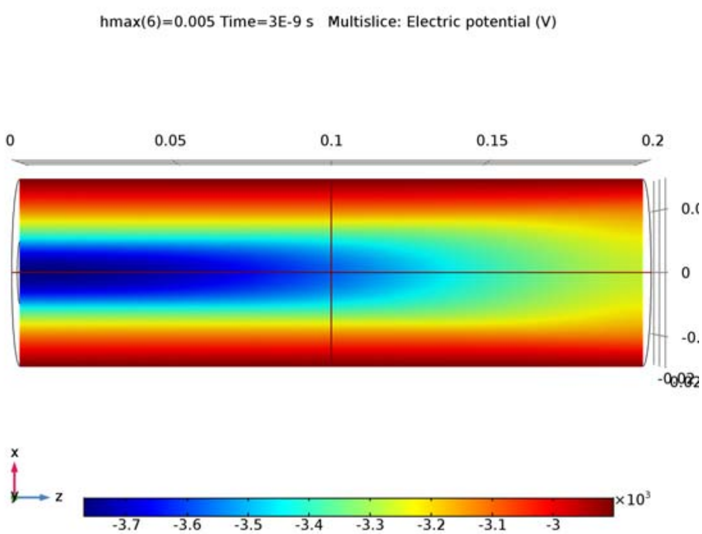


Figure 2: Electric potential in the relativistic beam. The magnitude of the potential is greatest at the beam waist.

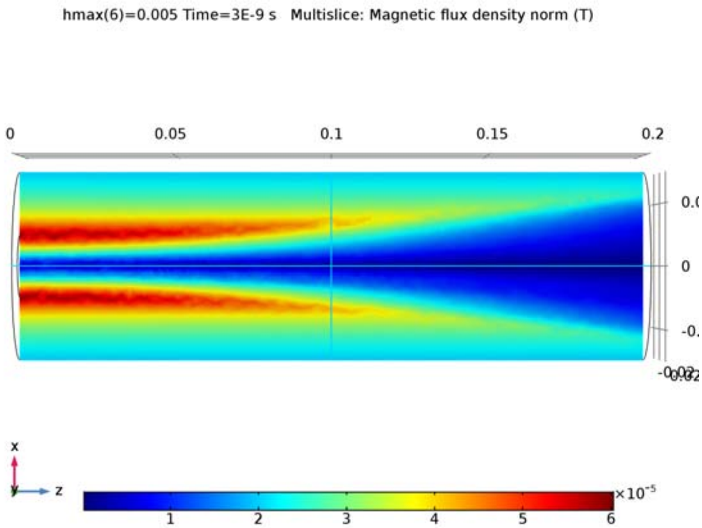


Figure 3: Magnetic flux density norm in the beam.

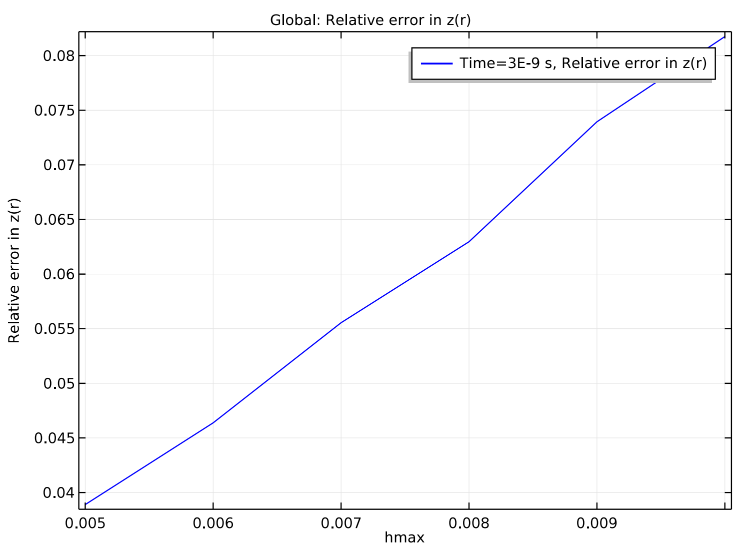


Figure 4: Relative error in z(r), the axial position as a function of beam radius.

Reference

1. S. Humphries, *Charged Particle Beams*, Dover Publications, New York, 2013.

Application Library path: `ACDC_Module/Particle_Tracing/electron_beam_divergence_relativistic`

Model 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>Particle Tracing>Particle Field Interaction, Relativistic**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Bidirectionally Coupled Particle Tracing**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

To save time, the parameters can be loaded from a file.

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

3 Click **Load from File**.

4 Browse to the application's Application Libraries folder and double-click the file `electron_beam_divergence_relativistic_parameters.txt`.

GEOMETRY 1

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r2`.

4 In the **Height** text field, type `L`.

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`tlayer`

Work Plane 1 (wp1)

1 Right-click **Cylinder 1 (cyl1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Work Plane**.

3 In the **Settings** window for Work Plane, locate the **Plane Definition** section.

4 From the **Plane type** list, choose **Face parallel**.

5 Find the **Planar face** subsection. Select the **Active** toggle button.

6 On the object **cyl1**, select Boundary 10 only.

7 Click **Show Work Plane**.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `r0beam`.

4 Right-click **Circle 1 (c1)** and choose **Build Selected**.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

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

2 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permeability	mur	1	1	Basic
Relative permittivity	epsilonr	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
qr	sqrt(qx^2+qy^2)	m	Radial distance from beam axis
qrmax	cpt.cptmaxop1(qr)	m	Beam radius
z_avg	cpt.cptaveop1(qz)	m	Average z-coordinate
chi	qrmax/at(0,qrmax)		Ratio of beam radius to waist radius

To model a drifting electron beam propagating in a large, open area, surround the modeling domain with an **Infinite Element Domain**. This results in appropriate boundary conditions for the electric potential.

Infinite Element Domain 1 (ie1)

1 On the **Definitions** toolbar, click **Infinite Element Domain**.

2 In the **Settings** window for Infinite Element Domain, locate the **Geometry** section.

3 From the **Type** list, choose **Cylindrical**.

4 Select Domains 1, 2, 4, and 5 only.

ELECTROSTATICS (ES)

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 Select Boundaries 2, 3, 14, and 22 only.

CHARGED PARTICLE TRACING (CPT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Charged Particle Tracing (cpt)**.

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 12 only.

3 In the **Settings** window for Inlet, locate the **Release Current Magnitude** section.

4 In the I text field, type `Ibeam`.

5 Locate the **Initial Position** section. From the **Initial position** list, choose **Density**.

6 In the N text field, type `1000`.

7 Locate the **Initial Velocity** section. Specify the $\mathbf{v}_0$ vector as

0	x
0	y
v0beam	z

Electric Force 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Charged Particle Tracing (cpt)** click **Electric Force 1**.

2 In the **Settings** window for Electric Force, locate the **Electric Force** section.

3 From the $\mathbf{E}$ list, choose **Electric field (es/ccn1)**.

4 Locate the **Advanced Settings** section. Select the **Use piecewise polynomial recovery on field** check box.

Magnetic Force 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Charged Particle Tracing (cpt)** click **Magnetic Force 1**.

2 In the **Settings** window for Magnetic Force, locate the **Magnetic Force** section.

3 From the $\mathbf{B}$ list, choose **Magnetic flux density (mf/al1)**.

4 Locate the **Advanced Settings** section. Select the **Use piecewise polynomial recovery on field** check box.

MESH 1

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

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

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

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, locate the **Domain Selection** section.

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

4 Select Domain 3 only.

Size 1

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

2 In the **Settings** window for Size, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

5 In the associated text field, type `hmax`.

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

Swept 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Swept 1** and choose **Distribution**.

Distribution 1

In the **Settings** window for Distribution, click **Build All**.

STUDY 1

Use a **Parametric Sweep** to confirm that a finer mesh results in closer agreement with the expected beam envelope size.

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
hmax	`range(0.01,-0.001,0.005)`	

The smallest mesh size requires about 3.5 GB RAM.

Step 1: Bidirectionally Coupled Particle Tracing

1 In the **Model Builder** window, under **Study 1** click **Step 1: Bidirectionally Coupled Particle Tracing**.
2 In the **Settings** window for Bidirectionally Coupled Particle Tracing, locate the **Study Settings** section.
3 In the **Times** text field, type `range(0,1.0e-10,3e-9)`.
4 Select the **Relative tolerance** check box.
5 In the associated text field, type `1.0E-5`.
6 Locate the **Iterations** section. In the **Number of iterations** text field, type `3`.
7 On the **Study** toolbar, click **Compute**.

RESULTS

Plot the trajectories of the electrons, using a **Color Expression** to observe their radial displacement over time.

Particle Trajectories 1

1 In the **Model Builder** window, expand the **Results>Particle Trajectories (cpt)** node, then click **Particle Trajectories 1**.
2 In the **Settings** window for Particle Trajectories, locate the **Coloring and Style** section.
3 Find the **Line style** subsection. From the **Type** list, choose **Line**.

Color Expression 1

1 In the **Model Builder** window, expand the **Particle Trajectories 1** node, then click **Color Expression 1**.
2 In the **Settings** window for Color Expression, locate the **Expression** section.
3 In the **Expression** text field, type `qr-at(0,qr)`.
4 On the **Particle Trajectories (cpt)** toolbar, click **Plot**.
5 Click the **Zoom Extents** button on the **Graphics** toolbar. This plot should look like Figure 1.

Data Sets

The electric potential and magnetic flux density can be visualized more clearly by excluding the selection of the **Infinite Element Domain** from the **Study 1/Parametric Solutions 1 (sol2)** data set.

Study 1/Parametric Solutions 1 (sol2)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Parametric Solutions 1 (sol2)**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 Select Domain 3 only.

Electric Potential (es)

1 In the **Model Builder** window, under **Results** click **Electric Potential (es)**.

2 In the **Settings** window for 3D Plot Group, click to expand the **Color legend** section.

3 Locate the **Color Legend** section. From the **Position** list, choose **Bottom**.

4 Click the **Go to ZX View** button on the **Graphics** toolbar. This plot should look like Figure 2.

Magnetic Flux Density Norm (mf)

1 In the **Model Builder** window, under **Results** click **Magnetic Flux Density Norm (mf)**.

2 In the **Settings** window for 3D Plot Group, click to expand the **Color legend** section.

3 Locate the **Color Legend** section. From the **Position** list, choose **Bottom**. This plot should look like Figure 3.

For each mesh size, compare the beam envelope shape to the analytical solution for a paraxial, nonrelativistic beam.

1D Plot Group 4

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for 1D Plot Group, type `Relative Error in Longitudinal Beam Displacement` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 From the **Time selection** list, choose **Last**.

Global 1

1 On the **Relative Error in Longitudinal Beam Displacement** toolbar, click **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
`abs((r0beam/sqrt(2*K)*integrate(1/sqrt(log(s)),s,1+eps,chi)-z_avg)/z_avg)`		`Relative error in z(r)`

This expression is Equation 2, where the first argument of the `integrate` operator is the integrand of Equation 4. The lower limit of integration is increased by the floating point relative accuracy `eps` (machine epsilon, 2^{-52} or about 2.2204×10^{-16}) to avoid division by zero during numerical integration.

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **hmax**.

5 On the **Relative Error in Longitudinal Beam Displacement** toolbar, click **Plot**. This plot should look like Figure 4.

Generator in 2D

Introduction

This example shows how the circular motion of a rotor with permanent magnets generates an induced EMF in a stator winding. The generated voltage is calculated as a function of time during the rotation. The model also shows the influence on the voltage from material parameters, rotation velocity, and number of turns in the winding.

The center of the rotor consists of annealed medium carbon steel, which is a material with a high relative permeability. The center is surrounded with several blocks of a permanent magnet made of samarium and cobalt, creating a strong magnetic field. The stator is made of the same permeable material as the center of the rotor, confining the field in closed loops through the winding. The winding is wound around the stator poles. Figure 1 shows the generator with part of the stator sliced in order to show the winding and the rotor.

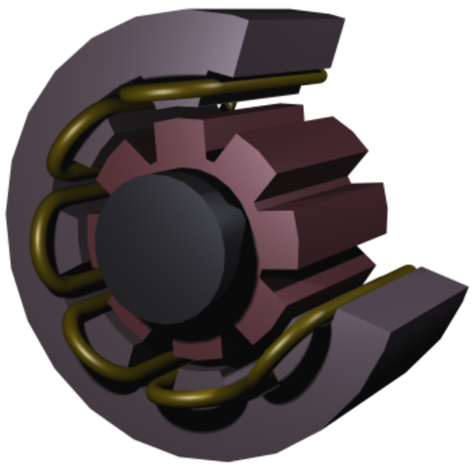


Figure 1: Drawing of a generator showing how the rotor, stator, and stator winding are constructed. The winding is also connected between the loops, interacting to give the highest possible voltage.

Modeling in COMSOL Multiphysics

The COMSOL Multiphysics model of the generator is a time-dependent 2D problem on a cross section through the generator. This is a true time-dependent model where the motion of the magnetic sources in the rotor is accounted for in the boundary condition between the stator and rotor geometries. Thus, there is no Lorentz term in the equation, resulting in the PDE

$$\sigma\frac{\partial \mathbf{A}}{\partial t}+\nabla\times\left(\frac{1}{\mu}\nabla\times\mathbf{A}\right) = 0$$

where the magnetic vector potential only has a z component.

Rotation is modeled using a ready-made physics interface for rotating machinery. The center part of the geometry, containing the rotor and part of the air-gap, rotates relative to the coordinate system of the stator. The rotor and the stator are imported as two separate geometry objects, so it is possible to use an assembly (see *Finalizing the Geometry* in the *COMSOL Multiphysics Reference Manual* for details). This has several advantages: the coupling between the rotor and the stator is done automatically, the parts are meshed independently, and it allows for a discontinuity in the vector potential at the interface between the two geometry objects (called slits). The rotor problem is solved in a rotating coordinate system where the rotor is fixed (the rotor frame), whereas the stator problem is solved in a coordinate system that is fixed with respect to the stator (the stator frame). An identity pair connecting the rotating rotor frame with the fixed stator frame is created between the rotor and the stator. The identity pair enforces continuity for the vector potential in the global fixed coordinate system (the stator frame).

The stator and center of the rotor are made of annealed medium-carbon steel (soft iron), which is implemented in COMSOL Multiphysics as an interpolation function of the B-H curve of the material; see Figure 2. The function can be used in the domain settings. Usually B-H curves are specified as $|\mathbf{B}|$ versus $|\mathbf{H}|$, but the rotating machinery, magnetic interface must have $|\mathbf{H}|$ versus $|\mathbf{B}|$. It is therefore important that the H-data is entered as $f(x)$-data of the interpolation function and the B-data entered as x-data. This relationship for |**H**| is predefined for the material Soft Iron (without losses) in the materials library that is shipped with the AC/DC Module.

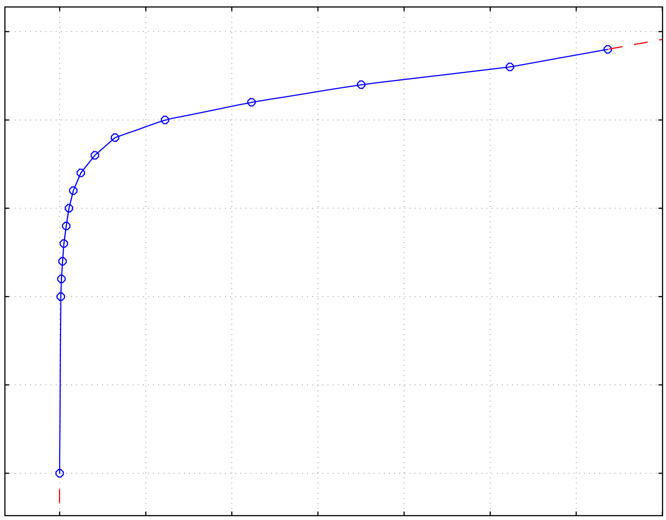


*Figure 2: The norm of the magnetic flux, |**B**|, versus the norm of the magnetic field, |**H**|, for the rotor and stator materials. The inverse of this curve is used in the calculation.*

The generated voltage is computed as the line integral of the electric field, **E**, along the winding. Because the winding sections are not connected in the 2D geometry, a proper line integral cannot be carried out. A simple approximation is to neglect the voltage contributions from the ends of the rotor, where the winding sections connect. The voltage is then obtained by taking the average z component of the **E** field for each winding cross-section, multiplying it by the axial length of the rotor, and taking the sum over all winding cross sections.

$$V_i = \mathrm{NN} \sum_{\text{windings}} \frac{L}{A} \int E_z dA$$

where L is the length of the generator in the third dimension, NN is the number of turns in the winding, and A is the total area of the winding cross-section.

Results and Discussion

The generated voltage in the rotor winding is a sinusoidal signal. At a rotation speed of 60 rpm the voltage has an amplitude slightly above 4 V for a single turn winding; see Figure 3.

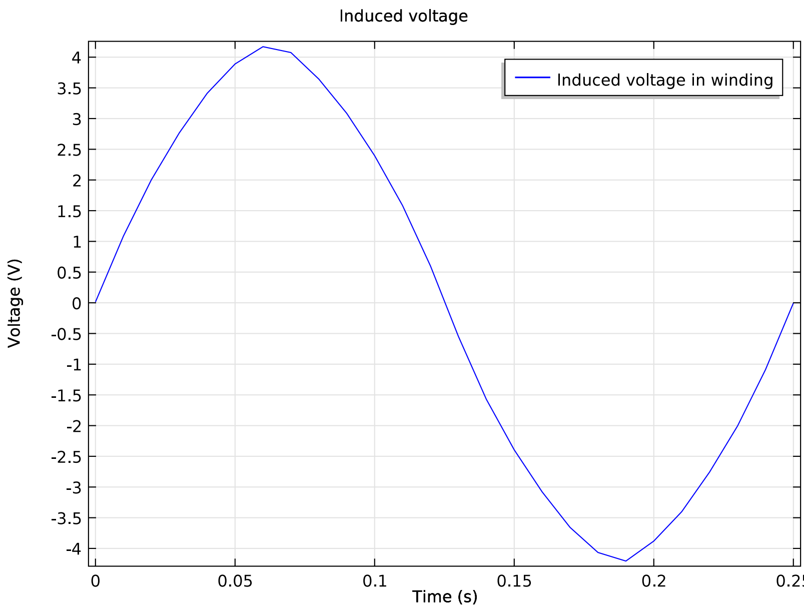


Figure 3: The generated voltage over one quarter of a revolution. This simulation used a single-turn winding.

The norm of the magnetic flux, $|\mathbf{B}|$, and the field lines of the $\mathbf{B}$ field are shown below in Figure 4 at time 0.20 s.

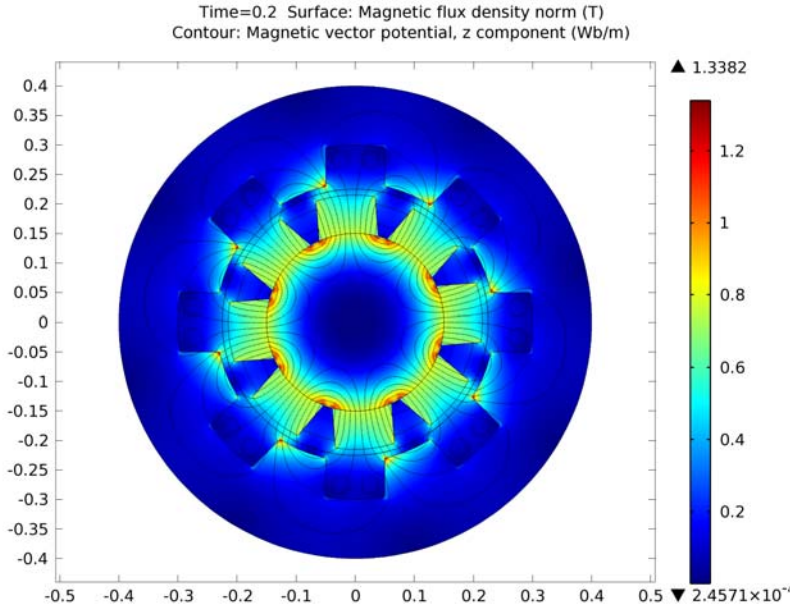


Figure 4: The norm and the field lines of the magnetic flux after 0.2 s of rotation. Note the brighter regions, which indicate the position of the permanent magnets in the rotor.

Application Library path: `ACDC_Module/Motors_and_Actuators/generator_2d`

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 **2D**.
2. In the **Select Physics** tree, select **AC/DC>Rotating Machinery, Magnetic (rmm)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1. On the **Home** toolbar, click **Parameters**.
2. In the **Settings** window for Parameters, locate the **Parameters** section.
3. In the table, enter the following settings:

Name	Expression	Value	Description
`L`	`0.4[m]`	0.4 m	`Length of generator`
`rpm`	`60[1/min]`	1 1/s	`Rotational speed of rotor`

GEOMETRY 1

Import the geometry of the generator cross section from an external CAD file.

Import 1 (imp1)

1. On the **Home** toolbar, click **Import**.
2. In the **Settings** window for Import, locate the **Import** section.
3. Click **Browse**.
4. Browse to the application's Application Libraries folder and double-click the file `generator_2d.mphbin`.
5. Click **Import**.

Form Union (fin)

The geometry you imported is composed by two objects, an inner part (corresponding to the rotor) and an outer part (the stator). Create an assembly from these objects.

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.
2 In the **Settings** window for Form Union/Assembly, locate the **Form Union/Assembly** section.
3 From the **Action** list, choose **Form an assembly**.
4 On the **Home** toolbar, click **Build All**.

   In this way, an identity pair connecting the shared boundaries has been automatically created.

DEFINITIONS

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
2 Select Domains 5–8 and 13–16 only.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
2 Select Domains 3, 4, 9–12, 17, and 18 only.

Integration 3 (intop3)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
2 Select Domain 8 only.

Cylindrical System 2 (sys2)

1 On the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.

   The cylindrical coordinate system you just added will be used to define the field of the permanent magnets.

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

ADD MATERIAL

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

2 In the tree, select **AC/DC>Soft Iron (without losses)**.

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

MATERIALS

Soft Iron (without losses) (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Soft Iron (without losses) (mat2)**.

2 Select Domains 2 and 28 only.

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

ROTATING MACHINERY, MAGNETIC (RMM)

Apply a Prescribed Rotational Velocity feature to the rotor, and specify its rotational velocity using the parameter `rpm`.

Prescribed Rotational Velocity 1

1 On the **Physics** toolbar, click **Domains** and choose **Prescribed Rotational Velocity**.

2 Select Domains 19–28 only.

3 In the **Settings** window for Prescribed Rotational Velocity, locate the **Prescribed Rotational Velocity** section.

4 In the rps text field, type `rpm`.

The permanent magnets on the rotor have a radial field. Use the cylindrical coordinate system specified earlier to define it.

Ampère's Law 2

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domains 20, 23, 24, and 27 only.

3 In the **Settings** window for Ampère's Law, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

5 Locate the **Magnetic Field** section. From the **Constitutive relation** list, choose **Remanent flux density**.

6 Specify the $\mathbf{B}_r$ vector as

0.84[T]	r

0	phi
0	a

7 Right-click **Ampère's Law 2** and choose **Rename**.

8 In the **Rename Ampère's Law** dialog box, type `Permanent Magnets Outward` in the **New label** text field.

9 Click **OK**.

The other four magnets are reversed.

Ampère's Law 3

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domains 21, 22, 25, and 26 only.

3 In the **Settings** window for Ampère's Law, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

5 Locate the **Magnetic Field** section. From the **Constitutive relation** list, choose **Remanent flux density**.

6 Specify the $\mathbf{B_r}$ vector as

-0.84[T]	r
0	phi
0	a

7 Right-click **Ampère's Law 3** and choose **Rename**.

8 In the **Rename Ampère's Law** dialog box, type `Permanent Magnets Inward` in the **New label** text field.

9 Click **OK**.

Ampère's Law 4

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 In the **Settings** window for Ampère's Law, locate the **Magnetic Field** section.

3 From the **Constitutive relation** list, choose **HB curve**.

4 Select Domains 2 and 28 only.

5 Right-click **Ampère's Law 4** and choose **Rename**.

6 In the **Rename Ampère's Law** dialog box, type `Iron` in the **New label** text field.

7 Click **OK**.

The problem will be solved separately in the fixed frame of the stator and the rotating frame of the rotor. Apply a Continuity feature on the shared boundaries (that are connected by **Identity Pair 1**).

Continuity 1

1. On the **Physics** toolbar, in the Boundary section, click **Pairs** and choose **Continuity**.
2. In the **Settings** window for Continuity, locate the **Pair Selection** section.
3. In the **Pairs** list, select **Identity Pair 1 (ap1)**.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

Size

1. In the **Settings** window for Size, locate the **Element Size** section.
2. From the **Predefined** list, choose **Finer**.
3. Click the **Custom** button.
4. Locate the **Element Size Parameters** section. In the **Resolution of narrow regions** text field, type `2`.
5. Click **Build All**.

STUDY 1

Time Dependent

On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

Step 2: Time Dependent

1. In the **Settings** window for Time Dependent, locate the **Study Settings** section.
2. In the **Times** text field, type `range(0,0.01,0.25)`.

GLOBAL DEFINITIONS

Variables 1

1. On the **Home** toolbar, click **Variables** and choose **Global Variables**.
2. In the **Settings** window for Variables, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`A`	`comp1.intop3(1)`	m²	`Cross-sectional area of winding`
`Vi`	`comp1.intop1(L*rmm.Ez/A) -comp1.intop2(L*rmm.Ez/A)`		`Induced voltage in winding`

Some adjustments to the solver settings are required in order to obtain a more stable solution.

STUDY 1

Solution 1 (sol1)

1. On the **Study** toolbar, click **Show Default Solver**.
2. In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
3. In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** node, then click **Fully Coupled 1**.
4. In the **Settings** window for Fully Coupled, click to expand the **Results while solving** section.
5. Click to expand the **Method and termination** section. Locate the **Method and Termination** section. From the **Jacobian update** list, choose **On every iteration**.
6. In the **Tolerance factor** text field, type `1e-3`.
7. On the **Study** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density (rmm)

Now, plot the solution in the spatial frame (the stator's fixed frame) at time $t = 0.2$ s.

Study 1/Solution 1 (sol1)

1. In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Solution 1 (sol1)**.
2. In the **Settings** window for Solution, locate the **Solution** section.
3. From the **Frame** list, choose **Spatial (x, y, z)**.

Magnetic Flux Density (rmm)

1. In the **Model Builder** window, under **Results** click **Magnetic Flux Density (rmm)**.
2. In the **Settings** window for 2D Plot Group, locate the **Data** section.

3 From the **Time (s)** list, choose **0.2**.

Contour 1

1 In the **Model Builder** window, right-click **Magnetic Flux Density (rmm)** and choose **Contour**.

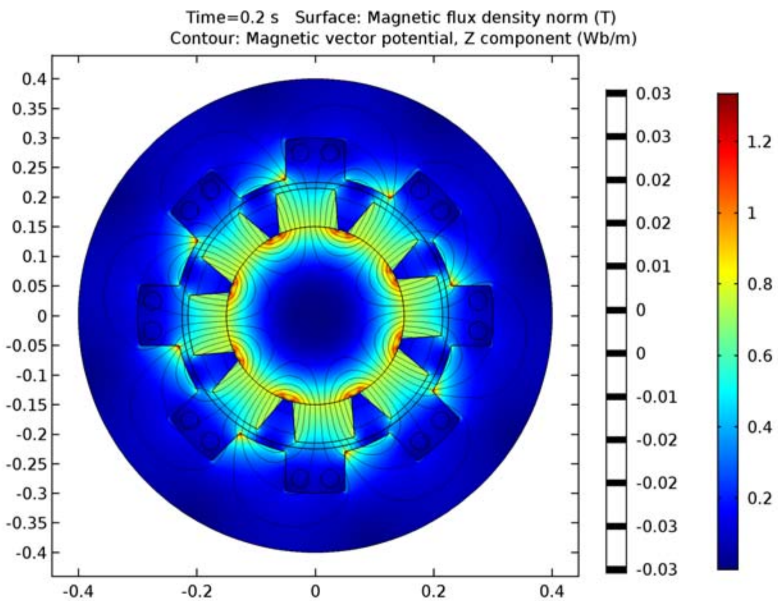
2 In the **Settings** window for Contour, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Rotating Machinery, Magnetic (Magnetic Fields)>Magnetic>Magnetic vector potential (Material)>Az - Magnetic vector potential, Z component**.

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

4 From the **Color** list, choose **Black**.

5 Locate the **Levels** section. In the **Total levels** text field, type 12.

6 On the **Magnetic Flux Density (rmm)** toolbar, click **Plot**.



The plot will show the rotor position at $t = 0.2$ s, the magnetic flux density norm and magnetic flux lines. Next, plot the induced EMF in a quarter of the cycle.

1D Plot Group 2

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

2 In the **Settings** window for 1D Plot Group, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Induced voltage`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `Time (s)`.

7 Select the **y-axis label** check box.

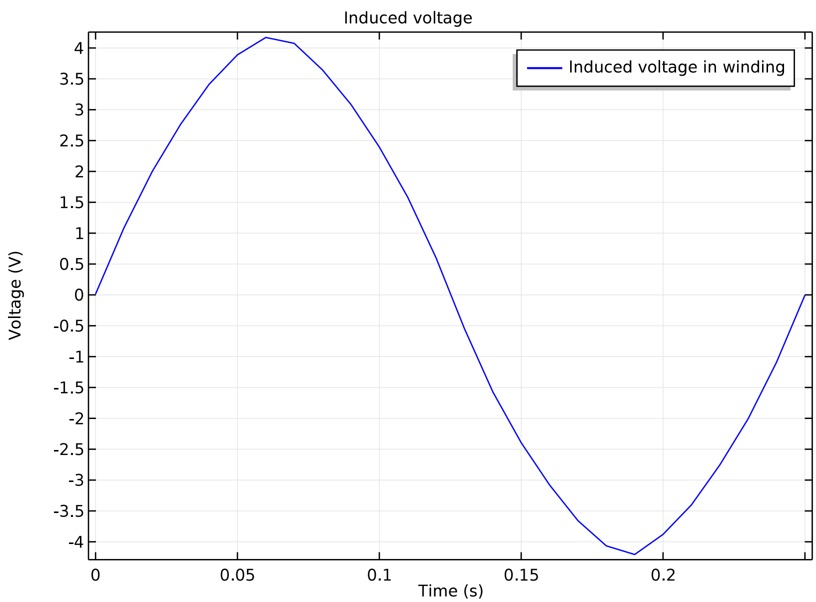
8 In the associated text field, type `Voltage (V)`.

Global 1

1 On the **1D Plot Group 2** toolbar, click **Global**.

2 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Global Definitions>Variables>Vi - Induced voltage in winding**.

3 On the **1D Plot Group 2** toolbar, click **Plot**.



The plot shows an induced EMF with an amplitude of about 4.2 V

Inductive Heating of a Copper Cylinder

Introduction

The induced currents in a copper cylinder produce heat, and when the temperature rises, the electric conductivity of the copper changes. Solving the heat transfer simultaneously with the field propagation is therefore crucial for an accurate description of this process.

The heating caused by the induced currents is called inductive heating. Generally heating due to currents is also called resistive heating or ohmic heating.

A challenge in induction heating is that the high current in the induction coils requires active cooling. This can be obtained by making the coil conductors hollow and circulating water inside. Even for rather modest flow rates, the coolant flow becomes highly turbulent which makes the heat transfer between conductor and fluid very efficient. This example illustrates a simplified way of modeling water cooling based on the assumption of turbulent flow and instantaneous mixing.

For mechanical support and electrical insulation, the cylinder and coil are embedded in FR4 composite material.

Model Definition

The system to be solved is given by

$$j\omega\sigma(T)\mathbf{A} + \nabla \times (\mu^{-1}\nabla \times \mathbf{A}) = 0$$

$$\rho C_p \frac{\partial T}{\partial t} - \nabla \cdot k\nabla T = Q(T, \mathbf{A})$$

where ρ is the density, C_p is the specific heat capacity, k is the thermal conductivity, and Q is the inductive heating.

The electric conductivity of copper, σ, is given by the expression

$$\sigma = \frac{1}{[\rho_0(1 + \alpha(T - T_0))]}$$

where ρ_0 is the resistivity at the reference temperature $T_0 = 293$ K, α is the temperature coefficient of the resistivity, and T is the actual temperature in the domain.

The time average of the inductive heating over one period, is given by

$$Q = \frac{1}{2}\sigma|\mathbf{E}|^2$$

The coil conductor is cooled by a turbulent water flow in an internal cooling channel. This is emulated by a combination of a high effective thermal conductivity and a homogenized out-of-plane convective loss term:

$$Q_c = \frac{\frac{dM}{dt}C_p(T_{in} - T)}{2\pi rA}$$

where $\frac{dM}{dt}$ is the water mass flow, T_{in} is the water inlet temperature, r is the radial coordinate and A is the cross-section area of the cooling channel.

Results and Discussion

The temperature after 10 h is shown in Figure 1. The average temperature of the copper cylinder has increased from 293 K to 346 K during this time. The current in the coil has an amplitude of 2 kA.

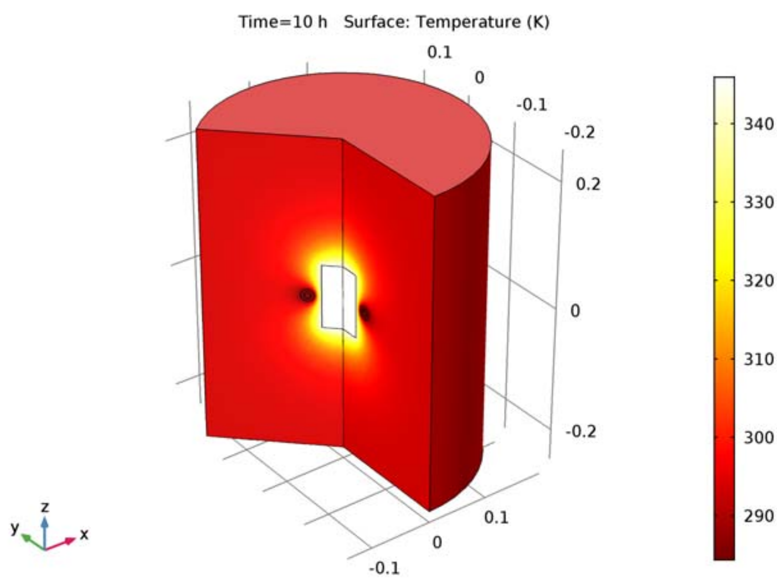


Figure 1: Temperature distribution after 10 h.

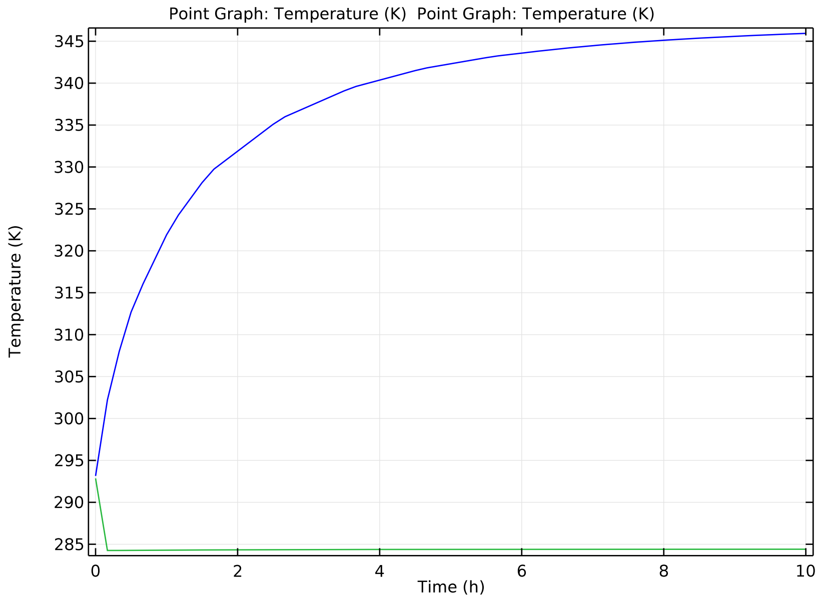


Figure 2: The plot shows the temperature evolution in the center of the copper cylinder and in the cooling channel.

Application Library path: `ACDC_Module/Electromagnetic_Heating/inductive_heating`

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 **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Heat Transfer>Electromagnetic Heating>Induction Heating**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics> Frequency-Transient**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
I0	2e3[A]	2000 A	Current
T0	293[K]	293 K	Reference temperature
r0	1.754e-8[ohm*m]	1.754E-8 Ω·m	Resistivity at T=T0
al	0.0039[1/K]	0.0039 1/K	Temperature coefficient
Rc	5[mm]	0.005 m	Cooling channel radius
Ac	pi*Rc^2	7.854E-5 m^2	Cooling channel x-section
Mt	1[kg/min]	0.01667 kg/s	Cooling water mass flow rate
Tin	10[degC]	283.2 K	Cooling water inlet temperature

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

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

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

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

Rectangle 2 (r2)

1 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

3 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

4 In the **Width** text field, type `0.03`.

5 In the **Height** text field, type `0.1`.

6 Locate the **Position** section. In the **z** text field, type `-0.05`.

Circle 1 (c1)

1 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

3 In the **Settings** window for Circle, locate the **Size and Shape** section.

4 In the **Radius** text field, type `0.01`.

5 Locate the **Position** section. In the **r** text field, type `0.05`.

6 Right-click **Circle 1 (c1)** and choose **Build Selected**.

Circle 2 (c2)

1 Right-click **Circle 1 (c1)** and choose **Duplicate**.

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

3 In the **Radius** text field, type `Rc`.

4 Right-click **Component 1 (comp1)>Geometry 1>Circle 2 (c2)** and choose **Build Selected**.

Form Union (fin)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

ADD MATERIAL

1 On 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>FR4 (Circuit Board)**.

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

ADD MATERIAL

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

2 In the tree, select **AC/DC>Copper**.

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

MATERIALS

Copper (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat2)**.

2 Select Domains 2 and 3 only.

3 In the **Model Builder** window, expand the **Copper (mat2)** node, then click **Linearized resistivity (ltr)**.

4 In the **Settings** window for Property Group, locate the **Output Properties and Model Inputs** section.

5 Find the **Output properties** subsection. In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Reference resistivity	rho0	r0	Ω·m	1x1
Resistivity temperature coefficient	alpha	a1	1/K	1x1
Reference temperature	Tref	T0	K	1x1

ADD MATERIAL

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

2 In the tree, select **Built-In>Water, liquid**.

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

MATERIALS

Water, liquid (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water, liquid (mat3)**.

2 Select Domain 4 only.

The built-in water material does not provide the electric permittivity and the magnetic permeability. Add those values.

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

4 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	80	1	Basic
Relative permeability	mur	1	1	Basic

Increase the thermal conductivity of the water to model the efficient heat transport in turbulent flow.

5 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	1e3	W/(m·K)	Basic

6 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MAGNETIC FIELDS (MF)

Add a separate **Ampère's Law** feature in the copper regions to specify a temperature-dependent resistivity.

Ampère's Law 2

1. On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.
2. Select Domains 2 and 3 only.
3. In the **Settings** window for Ampère's Law, locate the **Conduction Current** section.
4. From the σ list, choose **Linearized resistivity**.

Coil 1

1. On the **Physics** toolbar, click **Domains** and choose **Coil**.
2. Select Domain 3 only.
3. In the **Settings** window for Coil, locate the **Coil** section.
4. In the I_{coil} text field, type `I0`.

HEAT TRANSFER IN SOLIDS (HT)

Set up the Heat Transfer boundary conditions.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundaries 2, 7, and 9 only.

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type `T0`.

Heat Source 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 Select Domain 4 only.

3 In the **Settings** window for Heat Source, locate the **Heat Source** section.

4 In the Q_0 text field, type `Mt*ht.Cp*(Tin-T)/(2*pi*r*Ac)`.

MESH 1

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

2 In the **Settings** window for Mesh, click **Build All**.

STUDY 1

Step 1: Frequency-Transient

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Frequency-Transient**.

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

3 From the **Time unit** list, choose **h**.

4 Click **Range**.

5 In the **Range** dialog box, type `10[min]` in the **Step** text field.

6 In the **Stop** text field, type `10[h]`.

7 Click **Replace**.

8 In the **Settings** window for Frequency-Transient, locate the **Study Settings** section.

9 In the **Frequency** text field, type `500[Hz]`.

10 On the **Home** toolbar, click **Compute**.

RESULTS

Temperature, 3D (ht)

The revolution plot shows the temperature distribution after 10 hours; compare with Figure 1.

Create point data sets for plotting the temperature evolution in the copper cylinder and in the cooling channel.

Cut Point 2D 1

1. On the **Results** toolbar, click **Cut Point 2D**.
2. In the **Settings** window for Cut Point 2D, locate the **Point Data** section.
3. In the **r** text field, type `0`.
4. In the **z** text field, type `0`.

Cut Point 2D 2

1. Right-click **Cut Point 2D 1** and choose **Duplicate**.
2. In the **Settings** window for Cut Point 2D, locate the **Point Data** section.
3. In the **r** text field, type `0.05`.

1D Plot Group 5

On the **Results** toolbar, click **1D Plot Group**.

Point Graph 1

1. On the **1D Plot Group 5** toolbar, click **Point Graph**.
2. In the **Settings** window for Point Graph, locate the **Data** section.
3. From the **Data set** list, choose **Cut Point 2D 1**.
4. Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Heat Transfer in Solids>Temperature>T - Temperature**.

Point Graph 2

1. Right-click **Point Graph 1** and choose **Duplicate**.
2. In the **Settings** window for Point Graph, locate the **Data** section.
3. From the **Data set** list, choose **Cut Point 2D 2**.
4. On the **1D Plot Group 5** toolbar, click **Plot**.

   The plot shows the temperature evolution in the center of the copper cylinder and in the cooling channel; compare with Figure 2.

   Finish the modeling session by saving a representative model thumbnail.

Temperature, 3D (ht)

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

ROOT

1. In the **Model Builder** window, click the root node.

2 In the root nodes'**Settings** window, locate the **Presentation** section.

3 Find the **Thumbnail** subsection. Click **Set from Graphics Window**.

Created in COMSOL Multiphysics 5.2a

Modeling of a 3D Inductor

Introduction

Inductors are used in many applications for low-pass filtering or for impedance matching of predominantly capacitive loads. They are used in a wide frequency range from near static up to several MHz. An inductor usually has a magnetic core to increase the inductance while keeping its size small. The magnetic core also reduces the electromagnetic interference with other devices as the magnetic flux tends to stay within it. Because there are only crude analytical or empirical formulas available to calculate impedances, computer simulations or measurements are necessary during the design stage. In general, inductor modeling is more complex than modeling resistors and capacitors but similar principles apply. The model geometry is designed using an external CAD software, then it is imported into the AC/DC Module for static and frequency domain analysis. The inductor geometry is shown in Figure 1.

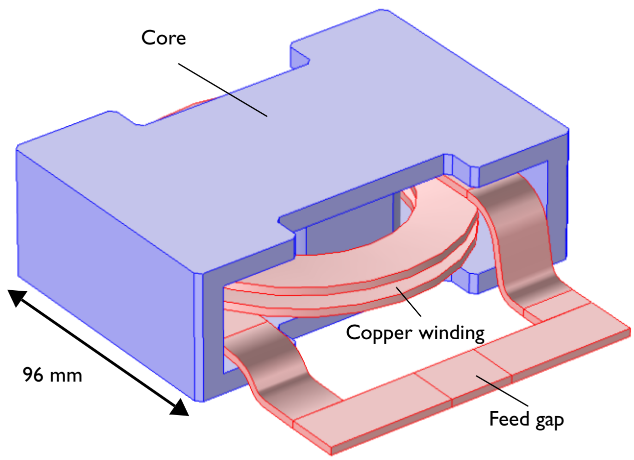


Figure 1: The inductor geometry.

First a magnetostatic simulation is performed to get the DC inductance. At low frequencies capacitive effects are negligible. Thus, the relevant equivalent circuit model is an ideal inductor in a series with an ideal resistor. The inductance and the resistance are both computed in the magnetostatic simulation. At a high frequency, capacitive effects and skin effect become significant and the equivalent circuit model involves connecting an ideal capacitor in parallel with the DC circuit. The skin effect modifies the current distribution in the winding so the resistance increases and the inductance also changes. The circuit parameters are obtained by analyzing the frequency dependent impedance obtained from a frequency domain simulation. In this tutorial, the AC analysis is done up to the point when the frequency dependent impedance is computed.

Model Definition

The application uses the **Magnetic Fields** interface which supports stationary, transient and frequency domain modeling. The following table lists the material properties used in this application:

MATERIAL PARAMETER	COPPER WINDING	CORE	AIR
σ	$5.998 \cdot 10^7$ S/m	0 S/m	0 S/m
ε_r	1	1	1
μ_r	1	10^3	1

The outer boundaries are set to the default magnetic insulation,

$$\mathbf{n} \times \mathbf{A} = \mathbf{0}$$

which from the inductive perspective is equivalent to a perfect electric conductor. In the magnetostatic analysis, the coil is modeled by a **Coil** feature which computes the current flow by means of a **Coil Geometry Analysis** preprocessing step, then applies a total current of 1 A. For the frequency domain analysis, instead a **Lumped Port** with a fixed a current of 1 A is applied to the feed gap.

Results and Discussion

The magnetostatic analysis yields an inductance of 0.11 mH and a DC resistance of 0.29 m·Ω. Figure 2 shows the magnetic flux density norm and the direction of the current flow.

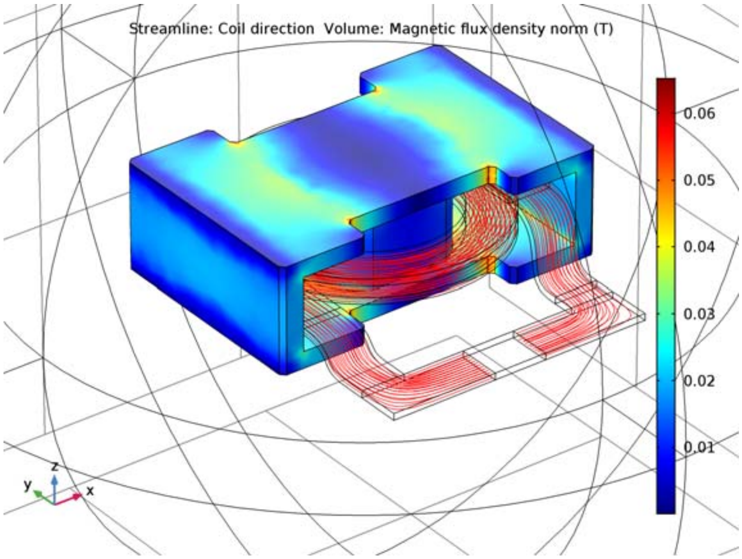


Figure 2: Magnetic flux density norm and current direction for the magnetostatic analysis.

In the static (DC) limit, the potential drop along the winding is purely resistive and could in principle be computed separately and before the magnetic flux density is computed. When increasing the frequency, inductive effects start to limit the current and skin effect makes it increasingly difficult to resolve the current distribution in the winding. At sufficiently high frequency, the current is mainly flowing in a thin layer near the conductor surface. When increasing the frequency further, capacitive effects come into play and current is flowing across the winding as displacement current density. When going through the resonance frequency, the device goes from behaving as an inductor to become predominantly capacitive. At the self resonance, the resistive losses peak due to the large internal currents. Figure 4 shows the surface current distribution at1 MHz. Typical for high frequency, the currents are displaced towards the edges of the conductor.

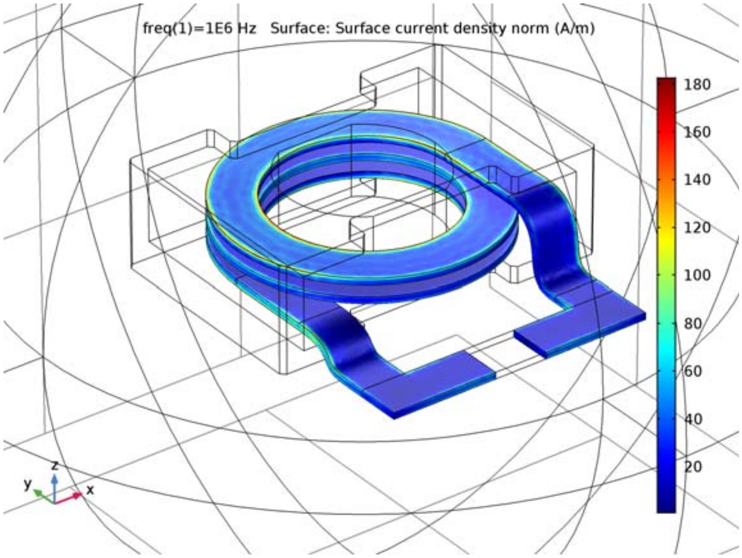


Figure 3: Surface current density at 1 MHz (below the resonance frequency).

Figure 4 shows how the resistive part of the coil impedance peaks at the resonance frequency near 6 MHz whereas Figure 5 shows how the reactive part of the coil impedance changes sign and goes from inductive to capacitive when passing through the resonance.

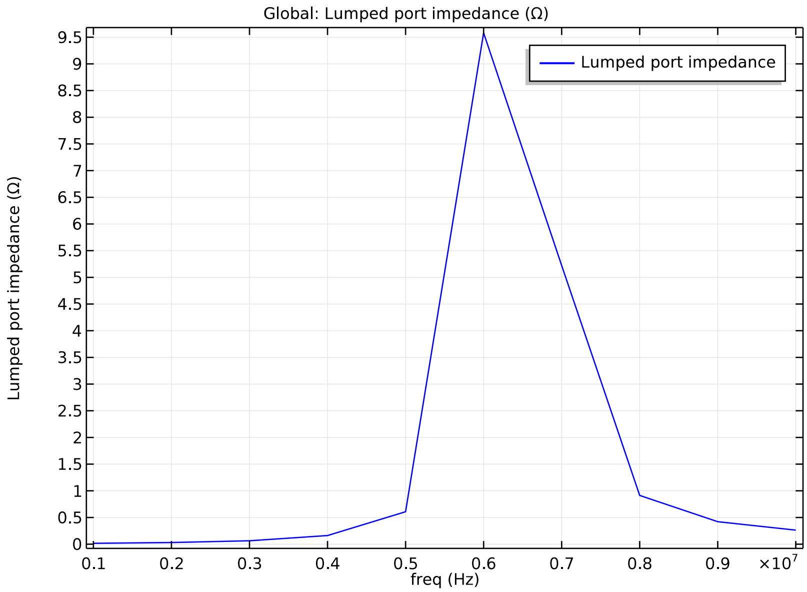


Figure 4: Real part of the electric potential distribution.

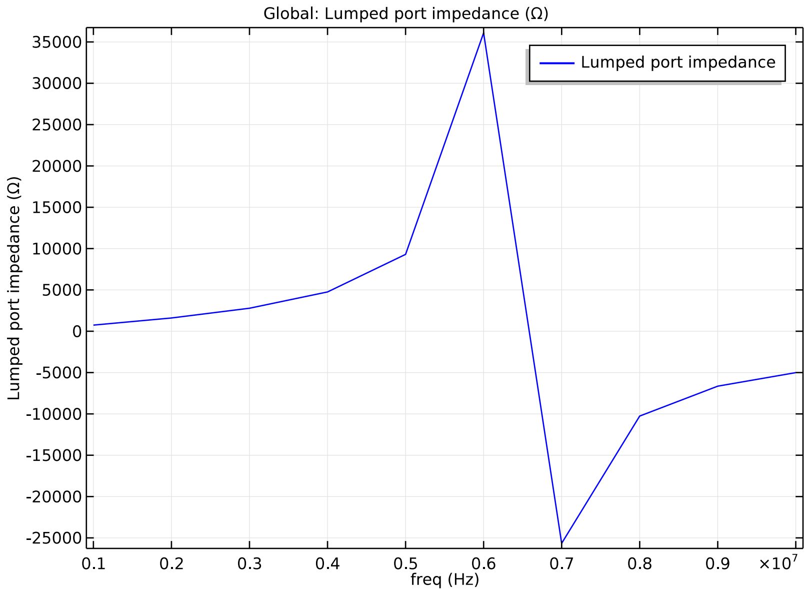


Figure 5: The reactive part of the coil impedance changes sign when passing through the resonance frequency, going from inductive to capacitive.

Application Library path: `ACDC_Module/Inductive_Devices_and_Coils/inductor_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>Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

The main geometry is imported from file. Air domains are typically not part of a CAD geometry so they usually have to be added later. For convenience three additional domains have been defined in the CAD file. These are used to define a narrow feed gap where an excitation can be applied.

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the application's Application Libraries folder and double-click the file `inductor_3d.mphbin`.

5 Click **Import**.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for Sphere, locate the **Size** section.

3 In the **Radius** text field, type `0.2`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.05

5 Click **Build All Objects**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for Form Union/Assembly, locate the **Form Union/Assembly** section.

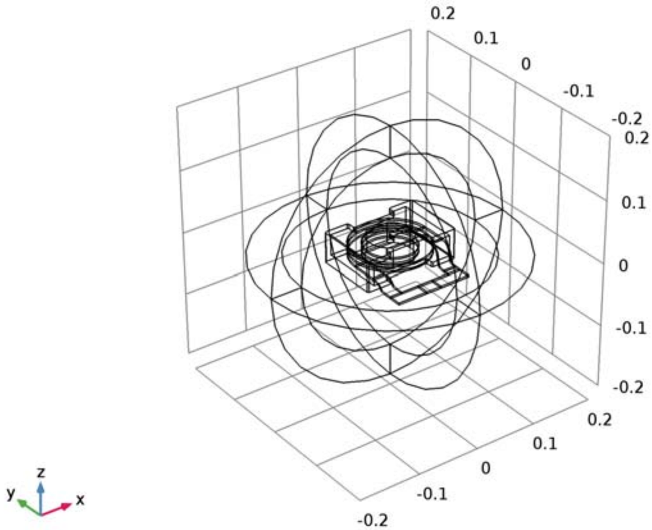
3 From the **Repair tolerance** list, choose **Relative**.

4 On the **Geometry** toolbar, click **Build All**.

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

6 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

The geometry should now look as in the figure below.



MATERIALS

Next, define selections to be used when setting up materials and physics. Start by defining the domain group for the inductor winding and continue by adding other useful selections.

DEFINITIONS

Explicit 1

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

2 Select Domains 7, 8, and 14 only.

3 Right-click **Explicit 1** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Winding` in the **New label** text field.

5 Click **OK**.

Explicit 2

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

2 Select Domain 9 only.

3 Right-click **Explicit 2** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Gap` in the **New label** text field.

5 Click **OK**.

Explicit 3

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

2 Select Domain 6 only.

3 Right-click **Explicit 3** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Core` in the **New label** text field.

5 Click **OK**.

Explicit 4

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

2 Select Domains 1–4 and 10–13 only.

3 Right-click **Explicit 4** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Infinite Elements` in the **New label** text field.

5 Click **OK**.

Explicit 5

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

2 Select Domains 1–6 and 9–13 only.

3 Right-click **Explicit 5** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Non-conducting` in the **New label** text field.

5 Click **OK**.

Explicit 6

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

2 Select Domains 5, 6, and 9 only.

3 Right-click **Explicit 6** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Non-conducting without IE` in the **New label** text field.

5 Click **OK**.

Use infinite elements to emulate an infinite open space surrounding the inductor.

Infinite Element Domain 1 (ie1)

1 On the **Definitions** toolbar, click **Infinite Element Domain**.

2. In the **Settings** window for Infinite Element Domain, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Infinite Elements**.
4. Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

   Now define the material settings.

ADD MATERIAL

1. On the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **AC/DC>Copper**.
4. Click **Add to Component** in the window toolbar.

MATERIALS

Copper (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat1)**.
2. In the **Settings** window for Material, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Winding**.

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 **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat2)**.
2. In the **Settings** window for Material, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Non-conducting**.

   The core material is not part of the material library so it is entered as a user-defined material.

Material 3 (mat3)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for Material, locate the **Geometric Entity Selection** section.

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

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

Property	Name	Value	Unit	Property group
Relative permeability	mur	1e3	1	Basic
Electrical conductivity	sigma	0	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

6 Right-click **Component 1 (comp1)>Materials>Material 3 (mat3)** and choose **Rename**.

7 In the **Rename Material** dialog box, type `Core` in the **New label** text field.

8 Click **OK**.

MAGNETIC FIELDS (MF)

Select Domains 1–8 and 10–14 only.

Coil 1

1 On the **Physics** toolbar, click **Domains** and choose **Coil**.

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

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

4 In the **Model Builder** window, expand the **Coil 1** node.

Input 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Magnetic Fields (mf)>Coil 1>Geometry Analysis 1** node, then click **Input 1**.

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `58` in the **Selection** text field.

6 Click **OK**.

Geometry Analysis 1

In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Fields (mf)>Coil 1** right-click **Geometry Analysis 1** and choose **Output**.

Output 1

1 In the **Settings** window for Output, locate the **Boundary Selection** section.

2 Click **Clear Selection**.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 79 in the **Selection** text field.

5 Click **OK**.

The steep radial scaling of the infinite elements region requires a swept mesh to maintain a reasonably good effective element quality. Turn on mesh control in the **Magnetic Fields** physics interface.

6 In the **Model Builder** window, click **Magnetic Fields (mf)**.

7 In the **Settings** window for Magnetic Fields, locate the **Physics-Controlled Mesh** section.

8 Select the **Enable** check box.

MESH 1

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

2 In the **Settings** window for Mesh, click **Build All**.

Add a **Coil Geometry Analysis** study step before the **Stationary** study to compute the direction of the current applied in the windings.

STUDY 1

Coil Geometry Analysis

On the **Study** toolbar, click **Study Steps** and choose **Other>Coil Geometry Analysis**.

Step 2: Coil Geometry Analysis

1 In the **Model Builder** window, under **Study 1** right-click **Step 2: Coil Geometry Analysis** and choose **Move Up**.

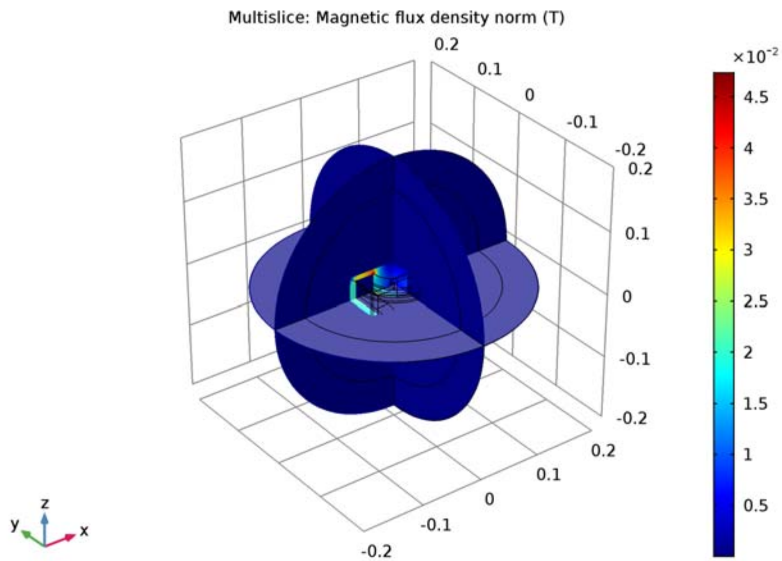
The magnetostatic model is now ready to solve.

2 On the **Study** toolbar, click **Compute**.

RESULTS

Study 1/Solution 1 (sol1)

The default plot group shows the magnetic flux density norm and helps in detecting possible modeling errors.



Better looking plots can be obtained by manipulating the data sets.

1 In the **Model Builder** window, expand the **Results>Data Sets** node.

2 Right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 From the **Selection** list, choose **Winding**.

Study 1/Solution 1 (1) (sol1)

In the **Model Builder** window, under **Results>Data Sets** right-click **Study 1/Solution 1 (1) (sol1)** and choose **Duplicate**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 From the **Selection** list, choose **Core**.

3D Plot Group 2

On the **Results** toolbar, click **3D Plot Group**.

Streamline 1

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

2 In the **Settings** window for Streamline, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields>Coil parameters>mf.coil1.eCoilx,...,mf.coil1.eCoilz - Coil direction**.

3 Select Boundary 58 only.

3D Plot Group 2

In the **Model Builder** window, under **Results** right-click **3D Plot Group 2** and choose **Volume**.

Volume 1

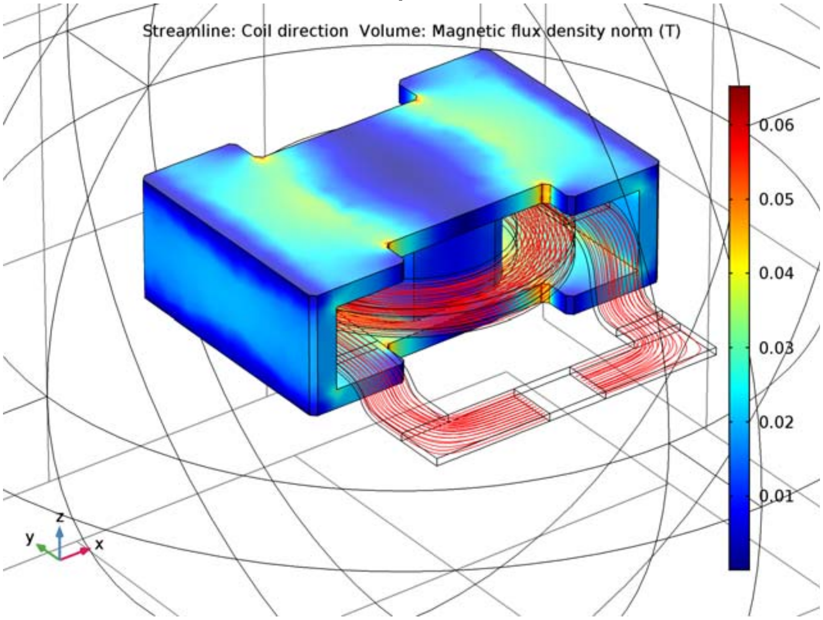
1 In the **Settings** window for Volume, locate the **Data** section.

2 From the **Data set** list, choose **Study 1/Solution 1 (4) (sol1)**.

3 On the **3D Plot Group 2** toolbar, click **Plot**.

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

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



Derived Values

Next, evaluate the coil inductance and resistance.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for Global Evaluation, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`mf.VCoil_1/mf.ICoil_1`	Ω	

4 Click **Evaluate**.

Global Evaluation 2

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for Global Evaluation, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`2*mf.intWm/mf.ICoil_1^2`	H	

4 Click **Evaluate**.

You should get about 0.11 mH and 0.29 mΩ respectively.

Now, solve the model without the infinite elements. This should make no difference as almost all of the magnetic flux resides inside the core region.

MAGNETIC FIELDS (MF)

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

2 Select Domains 5–8 and 14 only.

STUDY 1

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

RESULTS

Global Evaluation 1

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 1**.

2 In the **Settings** window for Global Evaluation, click **Evaluate**.

Global Evaluation 2

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 2**.

2 In the **Settings** window for Global Evaluation, click **Evaluate**.

COMPONENT 1 (COMP1)

Next, connect a simple circuit to the model.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **AC/DC>Electrical Circuit (cir)**.

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

5 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ELECTRICAL CIRCUIT (CIR)

Change the coil excitation so it can connect to the circuit.

MAGNETIC FIELDS (MF)

On the **Physics** toolbar, click **Electrical Circuit (cir)** and choose **Magnetic Fields (mf)**.

Coil 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Fields (mf)** click **Coil 1**.
2. In the **Settings** window for Coil, locate the **Coil** section.
3. From the **Coil excitation** list, choose **Circuit (current)**.

ELECTRICAL CIRCUIT (CIR)

On the **Physics** toolbar, click **Magnetic Fields (mf)** and choose **Electrical Circuit (cir)**.

Voltage Source 1

1. On the **Electrical Circuit** toolbar, click **Voltage Source**.
2. In the **Settings** window for Voltage Source, locate the **Node Connections** section.
3. In the table, enter the following settings:

Label	Node names
p	1
n	0

Resistor 1

1. On the **Electrical Circuit** toolbar, click **Resistor**.
2. In the **Settings** window for Resistor, locate the **Node Connections** section.
3. In the table, enter the following settings:

Label	Node names
p	1
n	2

4. Locate the **Device Parameters** section. In the R text field, type `100[mohm]`.

   There is a special feature for connecting the circuit to the finite elements model.

External I Vs. U 1

1. On the **Electrical Circuit** toolbar, click **External I Vs. U**.
2. In the **Settings** window for External I Vs. U, locate the **External Device** section.
3. From the V list, choose **Coil voltage (mf/coil1)**.

4 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	2
n	0

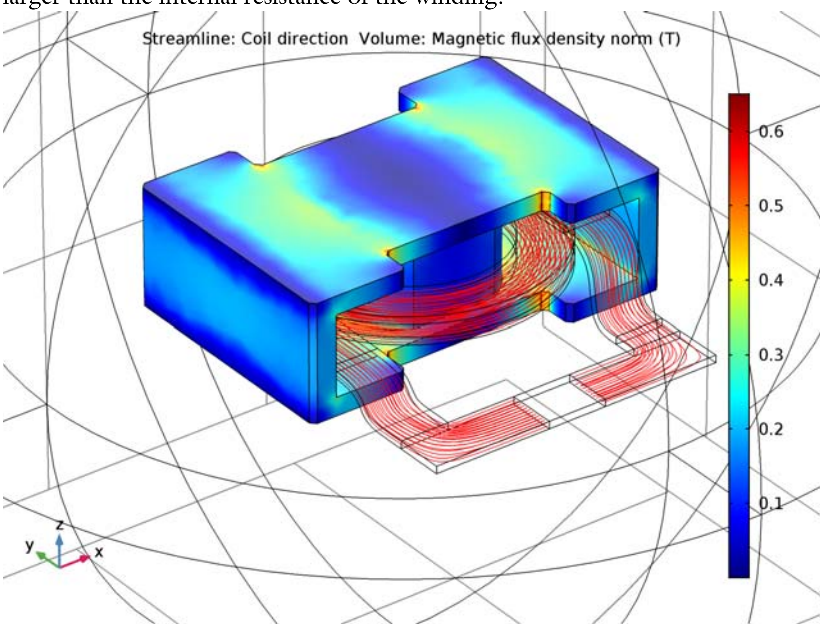
STUDY 1

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

RESULTS

3D Plot Group 2

The current is now limited to approximately 10 A by the external resistor, which is much larger than the internal resistance of the winding.



Now it is time to set up the model for computing the frequency dependent impedance. For this purpose, the **Magnetic Fields** interface is an alternative approach.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Electrical Circuit (cir)** interface.
4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Frequency Domain**.
5 Click **Add Study** in the window toolbar.
6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

DEFINITIONS

At high frequency the skin depth of the conductor cannot be resolved and the fields do not penetrate into the interior. Therefore, replace it by a lossy boundary condition. For this purpose add a selection and a surface material.

Explicit 7

1 On the **Definitions** toolbar, click **Explicit**.
2 Select Domains 7, 8, and 14 only.
3 In the **Settings** window for Explicit, locate the **Output Entities** section.
4 From the **Output entities** list, choose **Adjacent boundaries**.
5 Right-click **Explicit 7** and choose **Rename**.
6 In the **Rename Explicit** dialog box, type `Conductor Boundaries` in the **New label** text field.
7 Click **OK**.

ADD MATERIAL

1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **AC/DC>Copper**.
4 Click **Add to Component** in the window toolbar.
5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Copper 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 **Conductor Boundaries**.

MAGNETIC FIELDS (MF)

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

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

3 From the **Selection** list, choose **Non-conducting without IE**.

Apart from the surface loss in the copper conductor, there will also be loss in the core. For the static simulation this was not important but when approaching the resonance frequency of the coil it should be accounted for as otherwise the Q-factor becomes too high. The loss in the core is introduced as an effective loss tangent. For this purpose an extra equation/constitutive relation is required.

Ampère's Law 2

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 In the **Settings** window for Ampère's Law, locate the **Domain Selection** section.

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

4 Locate the **Electric Field** section. From the ε_r list, choose **User defined**. In the associated text field, type `1-5e-4*j`.

Impedance Boundary Condition 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Impedance Boundary Condition**.

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

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

Coil 1

The **Coil** feature no longer applies to an active domain so it needs to be disabled.

1 In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Fields (mf)** right-click **Coil 1** and choose **Disable**.

The electric potential is not available in this physics interface, so to excite the model you need to use a boundary feature that is more appropriate for high-frequency modeling.

Lumped Port 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Lumped Port**.

2 Select Boundaries 59–62 only.

The geometrical parameters of the boundary set need to be entered manually.

3 In the **Settings** window for Lumped Port, locate the **Lumped Port Properties** section.

4 From the **Type of lumped port** list, choose **User defined**.

5 In the h_{port} text field, type `0.024`.

6 In the w_{port} text field, type `0.046`.

7 Specify the $\mathbf{a}_{\mathrm{h}}$ vector as

1	x
0	y
0	z

8 From the **Terminal type** list, choose **Current**.

STUDY 2

Step 1: Frequency Domain

Set up a frequency sweep from 1 MHz to 10 MHz in steps of 1 MHz.

1 In the **Model Builder** window, under **Study 2** click **Step 1: Frequency Domain**.

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

3 Click **Range**.

4 In the **Range** dialog box, type `1e6` in the **Start** text field.

5 In the **Stop** text field, type `1e7`.

6 In the **Step** text field, type `1e6`.

7 Click **Replace**.

Near the resonance frequency the problem becomes ill-conditioned. If there was no loss, it would even become singular as the field solution would approach infinity. For a very high Q-factor, the iterative solver may fail to converge and then a direct solver must be used. Here it is sufficient to tweak the iterative solver to use a more robust preconditioner.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Study 2>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

4 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** node.

5 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** click **Iterative 1**.

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

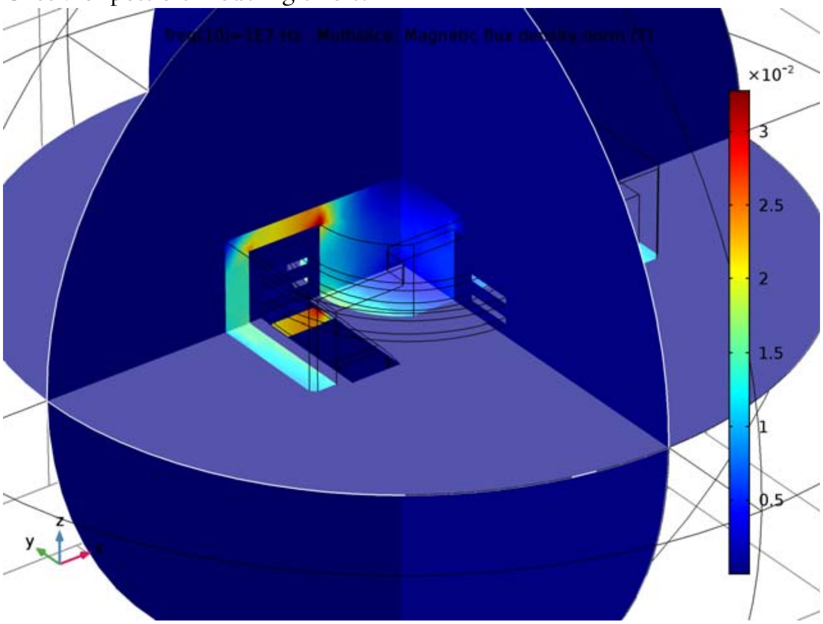
7 From the **Preconditioning** list, choose **Right**.

8 On the **Study** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density Norm (mf) 1

Check for possible modeling errors.



Proceed to look at the surface current distribution in the winding.

Study 2/Solution 3 (sol3)

In the **Model Builder** window, under **Results>Data Sets** right-click **Study 2/Solution 3 (sol3)** and choose **Duplicate**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 From the **Selection** list, choose **Conductor Boundaries**.

3D Plot Group 4

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for 3D Plot Group, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 3 (6) (sol3)**.

4 From the **Parameter value (freq (Hz))** list, choose **1E6**.

5 Right-click **3D Plot Group 4** and choose **Surface**.

Surface 1

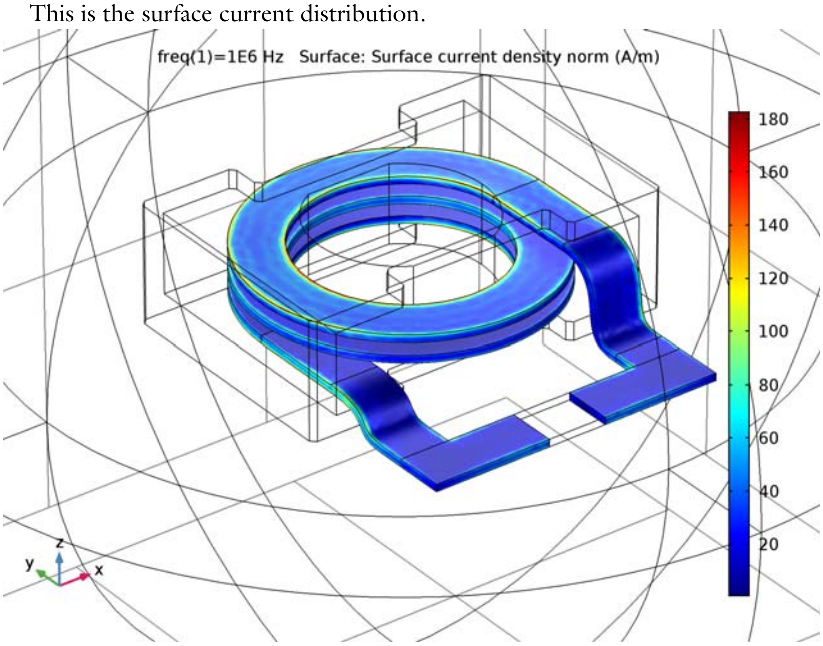
In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields>Currents and charge>mf.normJs - Surface current density norm**.

3D Plot Group 4

1 In the **Model Builder** window, under **Results** click **3D Plot Group 4**.

2 On the **3D Plot Group 4** toolbar, click **Plot**.

This is the surface current distribution.



Finish the modeling session by plotting the real and imaginary parts of the coil impedance.

1D Plot Group 5

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

2 In the **Settings** window for 1D Plot Group, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 3 (5) (sol3)**.

Global 1

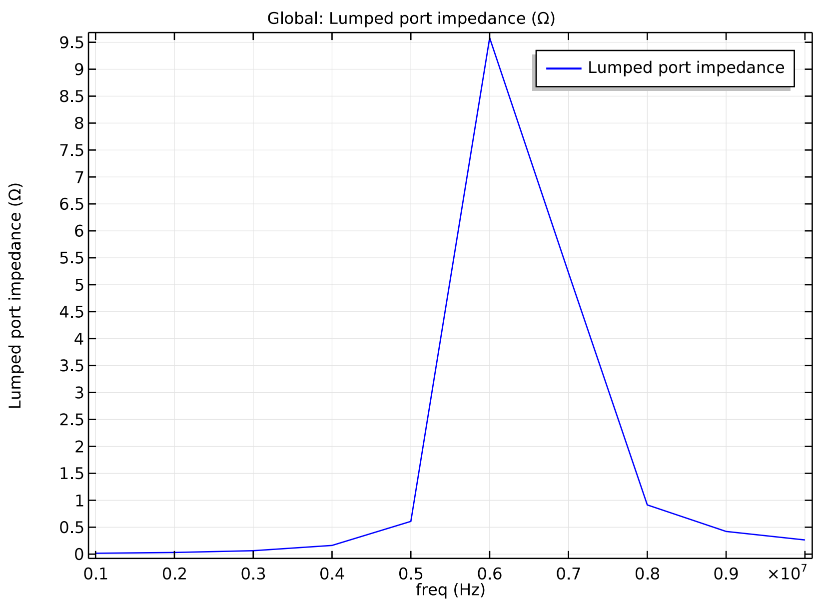
1 On the **1D Plot Group 5** toolbar, click **Global**.

2 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Magnetic Fields>Ports> mf.Zport_1 - Lumped port impedance**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`real(mf.Zport_1)`	Ω	`Lumped port impedance`

4 On the **1D Plot Group 5** toolbar, click **Plot**.



The resistive part of the coil impedance peaks at the resonance frequency.

1D Plot Group 6

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

2 In the **Settings** window for 1D Plot Group, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 3 (5) (sol3)**.

Global 1

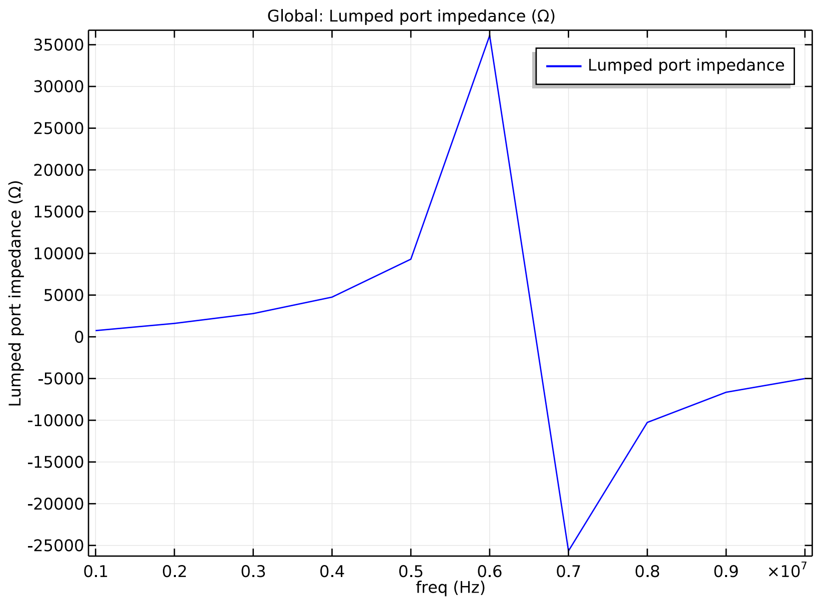
1 On the **1D Plot Group 6** toolbar, click **Global**.

2 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Magnetic Fields>Ports> mf.Zport_1 - Lumped port impedance**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`imag(mf.Zport_1)`	Ω	`Lumped port impedance`

4 On the **1D Plot Group 6** toolbar, click **Plot**.



The reactive part of the coil impedance changes sign when passing through the resonance frequency, going from inductive to capacitive.

Before saving the application, specify a plot to be used as a thumbnail.

Inductor in an Amplifier Circuit

Introduction

This example studies a finite element model of an inductor inserted into an electrical amplifier circuit.

Modern electronic systems are very complex and depend heavily on computer aided design in the development and manufacturing process. Common tools for such calculations are based on the SPICE format originally developed at Berkeley University (Ref. 1). The SPICE format consists of a standardized set of models for describing electrical devices—especially semiconductor devices such as transistors, diodes, and thyristors. SPICE also includes a simple, easy-to-read text format for circuit netlists and model parameter specifications. Although the netlist format is essentially the same as it was from the beginning, the set of models and model parameters constantly changes, with new models being added according to the latest achievements in semiconductor device development. When the devices are scaled down, new effects appear that have to be properly modeled. The new models are the result of ongoing research in device modeling.

When an engineer is designing a new electronic component, like a capacitor or an inductor, the SPICE parameters for that device are not known. They are either extracted from finite element tools such as COMSOL Multiphysics or from measurements on a prototype. To speed up the design process it can be convenient to include the finite element model in the SPICE circuit simulation, calculating the device behavior in an actual circuit.

This application takes a simple amplifier circuit and exchanges one of its components with a finite element model of an inductor with a magnetic core. COMSOL Multiphysics calculates the transient behavior of the entire system. Importing a SPICE circuit netlist brings in the circuit elements along with their model parameters and location in the circuit. All elements can be edited in COMSOL Multiphysics, and any pair of nodes can connect to the finite element model.

Model Definition

The inductor model uses the Magnetic Fields interface solving for the magnetic potential **A**:

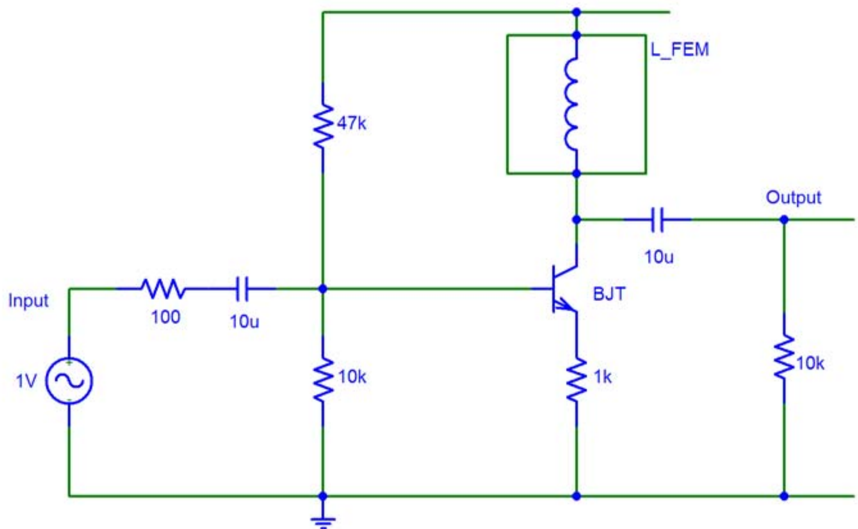
$$\sigma\frac{\partial \mathbf{A}}{\partial t} + \nabla \times (\mu_0^{-1}\mu_r^{-1}\nabla \times \mathbf{A}) = \mathbf{J}^e$$

where μ_0 is the permeability of vacuum, μ_r the relative permeability, and σ the electrical conductivity.

Because the coil has a large number of turns it is not efficient to model the separate wires and a homogenized formulation, where eddy currents within each wire are neglected, is used for the coil cross section. A dedicated coil feature is used for this purpose.

CONNECTION TO A SPICE CIRCUIT

The electrical circuit is a standard amplifier circuit with one bipolar transistor, biasing resistors, input filter, and output filter (see the figure below).



The input is a sine signal of 1 V and 10 kHz. The following listing shows the SPICE netlist for this circuit:

```
* BJT Amplifier circuit
.OPTIONS TNOM=27
.TEMP 27
Vin    1    0    sin(0 1 10kHz)
Vcc    4    0    15
Rg     1    2    100
Cin    2    3    10u
R1     4    3    47k
R2     3    0    10k
RE     7    0    1k
Cout   5    6    10u
Rl     6    0    10k
Q1     5    3    7    BJT
.MODEL BJT  NPN(Is=15f Ise=15f Isc=0 Bf=260 Br=6.1
+ Ikf=.3 Xtb=1.5 Ne=1.3 Nc=2 Rc=1  Rb=10 Eg=1.11
+ Cjc=7.5p Mjc=.35 Vjc=.75 Fc=.5 Cje=20p Mje=0.4 Vje=0.75
```

```
+ Vaf=75 Xtf=3 Xti=3)
.END
```

A dedicated circuit device is then added in COMSOL Multiphysics to connect the circuit with the inductor model. The model parameters of the transistor do not correspond to a real device, but the numbers are nevertheless chosen to be realistic.

The import of the SPICE netlist does not fully support the SPICE format; especially for the semiconductor device models it only supports a limited set of parameters. Supplying unsupported parameters results in those parameters not being used in the circuit model. For example, transit time capacitance and temperature dependence are not supported for the transistor model.

Results and Discussion

A first version of this application lets you compute the magnetic flux density distribution from a 1 A current through the inductor, without the circuit connection taken into consideration.

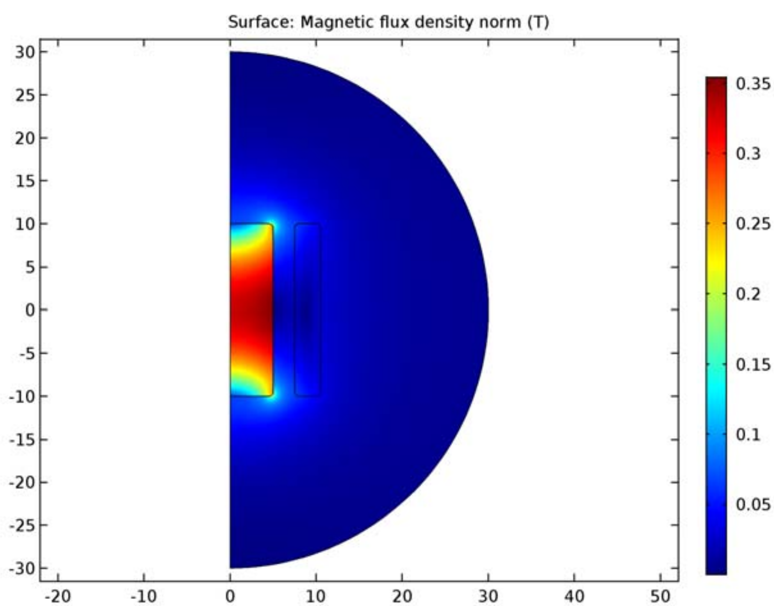


Figure 1: Magnetic flux density distribution as the coil is driven by a 1 A current source.

Biasing of an amplifier is often a complicated compromise, especially if you only use resistors. Adding an inductor as the collector impedance simplifies the biasing design, because the instantaneous voltage on the collector of the transistor can be higher than the supply voltage, which is not possible with resistors. Amplifiers using inductors can be quite narrow banded.

Before starting the transient simulation, proper initial conditions have to be calculated. For this application it is sufficient to ramp the supply voltage to 15 V with the nonlinear parametric solver. After the ramp, the DC bias conditions have been calculated properly, and you can use this solution as initial condition for the transient simulation.

Using a global variables plot, you can easily plot input signal, output signal, and inductor voltage in the same figure.

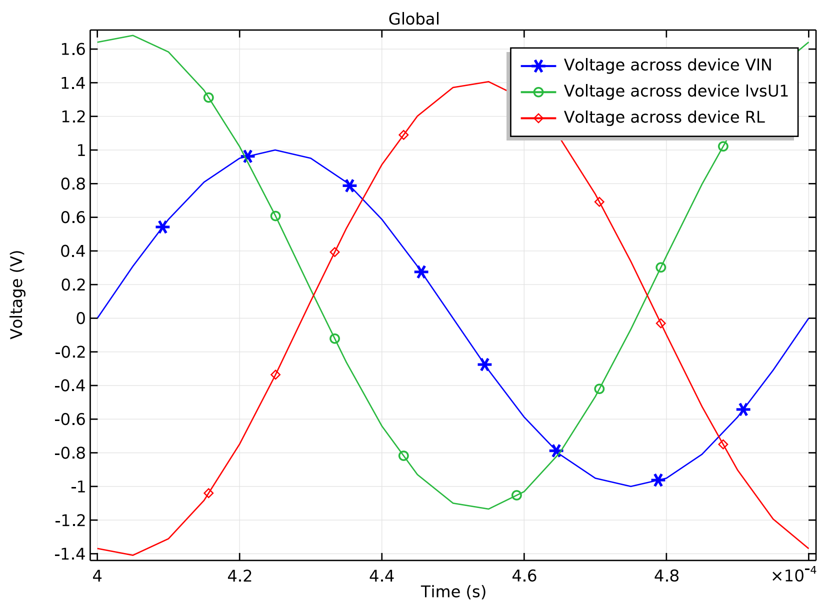


Figure 2: Input signal (cir.VIN_v), output signal (cir.RL_v), and inductor voltage (cir.X1_v) as functions of time.

The output signal is about 1.5 times the input signal in amplitude.

Reference

1. The SPICE home page, http://bwrc.eecs.berkeley.edu/Classes/IcBook/SPICE.

Application Library path: `ACDC_Module/Inductive_Devices_and_Coils/inductor_in_circuit`

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 **2D Axisymmetric**.
2 In the **Select Physics** tree, select **AC/DC>Magnetic Fields (mf)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Stationary**.
6 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for Geometry, locate the **Units** section.
3 From the **Length unit** list, choose **mm**.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
2 In the **Settings** window for Circle, locate the **Size and Shape** section.
3 In the **Radius** text field, type `30`.

Rectangle 1 (r1)

1 Right-click **Circle 1 (c1)** and choose **Build Selected**.
2 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
3 In the **Settings** window for Rectangle, locate the **Size and Shape** section.
4 In the **Width** text field, type `40`.
5 In the **Height** text field, type `80`.

6 Locate the **Position** section. In the **z** text field, type -40.

Intersection 1 (int1)

1 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

3 Select both the circle and the rectangle.

4 In the **Model Builder** window, right-click **Intersection 1 (int1)** and choose **Build Selected**.

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

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type 5.

4 In the **Height** text field, type 20.

5 Locate the **Position** section. In the **z** text field, type -10.

Rectangle 3 (r3)

1 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

3 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

4 In the **Width** text field, type 3.

5 In the **Height** text field, type 20.

6 Locate the **Position** section. In the **r** text field, type 7.5.

7 In the **z** text field, type -10.

Fillet 1 (fil1)

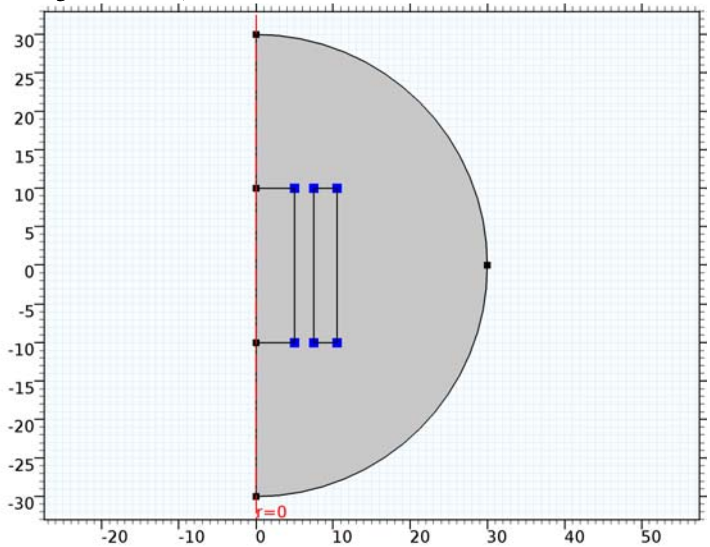
1 Right-click **Rectangle 3 (r3)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Fillet**.

Next, select all six points in the internal of the geometry as follows:

3 Click the **Select Box** button on the **Graphics** toolbar.

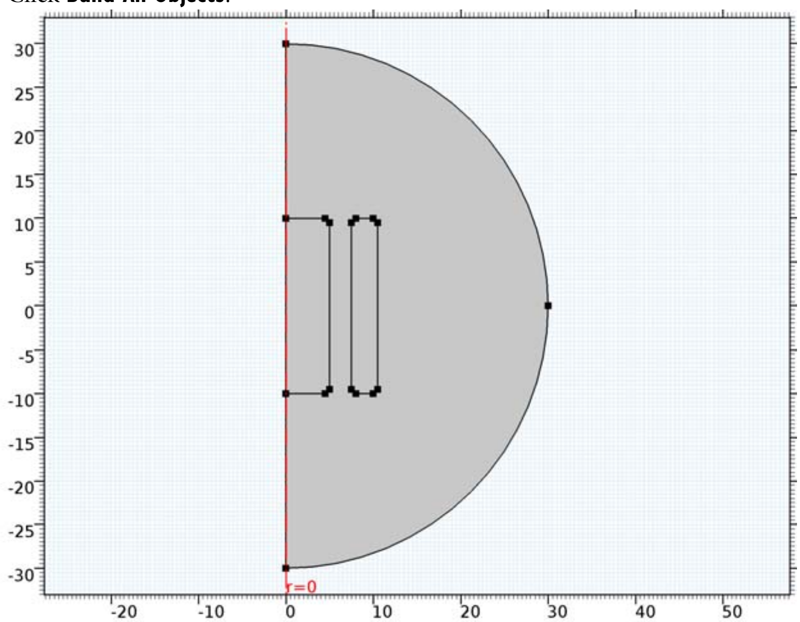
4 Using the mouse, enclose the internal vertices in a box to select them.



5 In the **Settings** window for Fillet, locate the **Radius** section.

6 In the **Radius** text field, type `0.5`.

7 Click **Build All Objects**.



GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
t	0[s]	0 s	Time for stationary solution
N	1e3	1000	Coil turns
freq	10[kHz]	10000 Hz	Frequency
d_coil	0.1[mm]	1E-4 m	Coil wire diameter
sigma_coil	5e7[S/m]	5E7 S/m	Wire conductivity
Vappl	15[V]	15 V	Supply voltage

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

ADD MATERIAL

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

2 In the tree, select **AC/DC>Soft Iron (without losses)**.

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

MATERIALS

Soft Iron (without losses) (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Soft Iron (without losses) (mat2)**.

2 Select Domain 2 only.

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

This leaves your model with material data for soft iron in the core and air elsewhere. Note that the behavior of the coil is determined by the applied current and the resulting voltage.

MAGNETIC FIELDS (MF)

First, give the solved-for magnetic potential an initial value with a nonzero gradient. This helps the nonlinear solver avoid an otherwise singular linearization before it takes the first step.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Fields (mf)** click **Initial Values 1**.

2 In the **Settings** window for Initial Values, locate the **Initial Values** section.

3 Specify the **A** vector as

0	r
1[uWb/m^2]*r	phi
0	z

Here, the prefix `u` in `uWb` stands for micro (μWb).

Next, set up the coil.

Ampère's Law 2

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domain 2 only.

3 In the **Settings** window for Ampère's Law, locate the **Magnetic Field** section.

4 From the **Constitutive relation** list, choose **HB curve**.

Coil 1

1 On the **Physics** toolbar, click **Domains** and choose **Coil**.

2 Select Domain 3 only.

3 In the **Settings** window for Coil, locate the **Coil** section.

4 From the **Conductor model** list, choose **Homogenized multi-turn**.

5 Locate the **Homogenized Multi-Turn Conductor** section. In the σ_{coil} text field, type `sigma_coil`.

6 In the N text field, type `N`.

7 From the **Coil wire cross-section area** list, choose **From round wire diameter**.

8 In the d_{coil} text field, type `d_coil`.

In the I_{coil} text field, keep the default value of `1[A]`.

MESH 1

The steepest field gradients and consequently the most important challenges to the convergence of this model are expected to occur in the vicinity of the fillets. You can increase the accuracy and help the solver by using a high resolution of narrow regions.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

2 In the **Settings** window for Size, click to expand the **Element size parameters** section.

3 Locate the **Element Size Parameters** section. In the **Resolution of narrow regions** text field, type 4.

4 Click **Build All**.

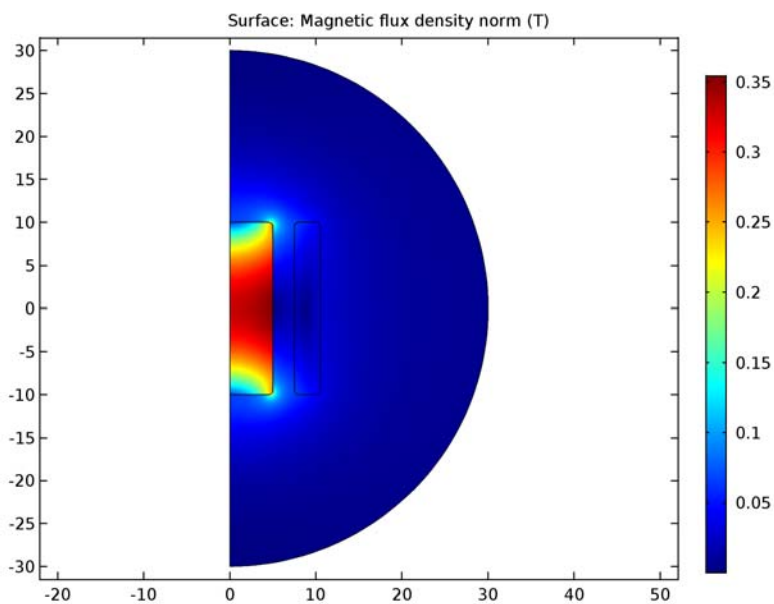
STUDY 1

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

RESULTS

Magnetic Flux Density Norm (mf)

The default plot shows the resulting magnetic flux density distribution from the applied 1 A current.



COMPONENT 1 (COMP1)

It is now time to add the circuit. Although you are eventually looking for transient results, the first solution step will use the stationary solver to ramp up the voltage from the voltage generator. You will therefore select a stationary study in the Model Wizard. First, prepare for the import by making the coil circuit-driven.

MAGNETIC FIELDS (MF)

To be able to keep the first version of the application fully intact, create a new **Coil** node for the circuit version of the application.

Coil 1

In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Fields (mf)** right-click **Coil 1** and choose **Duplicate**.

Coil 2

1 In the **Settings** window for Coil, locate the **Coil** section.
2 From the **Coil excitation** list, choose **Circuit (current)**.
3 In the **Coil name** text field, type 1.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2 Go to the **Add Physics** window.
3 In the tree, select **AC/DC>Electrical Circuit (cir)**.
4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.
5 Click **Add to Component** in the window toolbar.
6 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2 Go to the **Add Study** window.
3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.
4 Click **Add Study** in the window toolbar.
5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

ELECTRICAL CIRCUIT (CIR)

On the **Physics** toolbar, click **Magnetic Fields (mf)** and choose **Electrical Circuit (cir)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electrical Circuit (cir)** and choose **Import SPICE Netlist**.

2 Browse to the application's Application Libraries folder and double-click the file `amplifier.cir`.

Voltage Source VIN

The SPICE netlist is imported in the Circuit physics. In order to couple the amplifier with the inductor, an **External I vs U** feature must be connected between nodes 4 and 5.

External I Vs. U 1

1 On the **Electrical Circuit** toolbar, click **External I Vs. U**.

2 In the **Settings** window for External I Vs. U, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	4
n	5

Now prepare for the ramping of the voltage generator by changing the 15 V used in the voltage supply VCC to a parameter that the solver can sweep.

Voltage Source VCC

1 In the **Model Builder** window, under **Component 1 (comp1)>Electrical Circuit (cir)** click **Voltage Source VCC**.

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

3 In the V_{src} text field, type `Vappl`.

STUDY 1

Disable the new **Coil** node and the **Electrical Circuits** interface for **Study 1**. Conversely, you will disable the original node for the steps of **Study 2** shortly.

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

2 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Magnetic Fields (mf)>Coil 2**.

5 Click **Disable**.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Electrical Circuit (cir)**.

7 Click **Disable in Model**.

STUDY 2

The new study already contains a node for the initial stationary solution.

Time Dependent

On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

Step 2: Time Dependent

1 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

2 In the **Times** text field, type `range(0,5e-6,5e-4)`.

To get accurate results, you need to tighten the tolerances.

3 Select the **Relative tolerance** check box.

4 In the associated text field, type `1e-4`.

For the steps in this study, disable the original **Coil** node.

5 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Magnetic Fields (mf)>Coil 1**.

7 Click **Disable**.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

2 In the **Settings** window for Stationary, click to expand the **Study extensions** section.

3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Vappl	`range(1,15)`	

Using continuation rather than a parametric sweep lets you begin with a parametric solution and then use the result for the final parameter as the initial value for the time-dependent solver. In contrast, adding a parametric sweep would mean performing a transient solution for each parameter value.

6 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.

7 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Magnetic Fields (mf)>Coil 1**.

8 Click **Disable**.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Study 2>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Stationary Solver 1**.

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

5 In the **Relative tolerance** text field, type `1e-6`.

This model requires a somewhat tighter tolerance in the stationary solver than the default on account of the strong magnetic nonlinearity in the soft iron core material. A relative tolerance of 1×10^{-6} gives a very well-converged result, which is important for maintaining stability in the final time-dependent solver step.

6 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)** click **Time-Dependent Solver 1**.

7 In the **Settings** window for Time-Dependent Solver, click to expand the **Absolute tolerance** section.

8 Locate the **Absolute Tolerance** section. In the **Tolerance** text field, type `1e-6`.

ELECTRICAL CIRCUIT (CIR)

External I Vs. U 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electrical Circuit (cir)** click **External I Vs. U 1**.

2 In the **Settings** window for External I Vs. U, locate the **External Device** section.

3 From the V list, choose **Coil voltage (mf/coil2)**.

STUDY 2

On the **Study** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density Norm (mf) 1

The new default plot shows the flux density distribution at $t = 5\times10^{-4}$ s.

Follow the instructions below to plot the input and output signals as well as the inductor voltage versus time.

1D Plot Group 5

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

2 In the **Settings** window for 1D Plot Group, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 From the **Time selection** list, choose **From list**.

5 In the **Time** list, click and Shift-click to select all times between `4e-4` and `5e-4`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `Time (s)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Voltage (V)`.

Global 1

1 On the **1D Plot Group 5** toolbar, click **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
`cir.VIN_v`	`V`	`Voltage across device VIN`
`cir.IvsU1_v`	`V`	`Voltage across device IvsU1`
`cir.RL_v`	`V`	`Voltage across device RL`

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

1D Plot Group 5

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 5** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Voltages` in the **New label** text field.

3 Click **OK**.

The plot should now look like that in Figure 2.

Finish the modeling session by saving a representative model thumbnail.

ROOT

1 In the **Model Builder** window, click the root node.

2 In the root nodes'**Settings** window, locate the **Presentation** section.

3 Find the **Thumbnail** subsection. Click **Set from Graphics Window**.

Created in COMSOL Multiphysics 5.2a

Ion Funnel

Introduction

An electrodynamic ion funnel provides an efficient means of transferring ions from regions of high pressure to high vacuum. The ion funnel can couple devices which generally operate at pressures of different orders of magnitude, such as ion mobility spectrometers and mass spectrometers, allowing mixtures of ionized gases to be separated and analyzed while minimizing losses. For this reason, ion funnels can be used to improve instrument sensitivity in a wide variety of applications, such as the analysis of complex biological molecules or the detection of explosives.

This example uses the Charged Particle Tracing interface to model the movement of ions through an electrodynamic ion funnel. The Electric Force feature is used to apply DC and RF potentials to guide ions through the funnel, while the Elastic Collision Force feature is used to model collisions with background gas molecules.

Note: This application requires the Particle Tracing Module.

Model Definition

The ion funnel is a converging series of insulated ring-shaped electrodes, each subjected to an RF potential, with adjacent electrodes out of phase. The RF potential confines ions radially while a DC bias guides the ions toward successively narrower electrodes. The superposition of the DC and RF fields focuses the ions and offsets the effects of thermal dispersion and Coulombic repulsion.

The ion funnel contains a buffer gas of neutral argon at 1 Torr, which is assumed to follow a Maxwellian velocity distribution:

$$f(v_i) = \sqrt{\frac{m_p}{2\pi k_\mathrm{B} T_0}}\exp\left(\frac{m_p(v_i^2)}{2k_\mathrm{B}T_0}\right)$$

where

- v_i (SI unit: m/s) is the ith velocity component,
- m_p (SI unit: kg) is the molecular mass,
- T_0 (SI unit: K) is the temperature, and
- $k = 1.3806488 \cdot 10^{-23}$ J/K is the Boltzmann constant.

In this example, the background gas molecules have molar mass 0.04 kg/mol, for a molecular mass of $6.6422\cdot10^{-26}$ kg. The temperature is assumed to be 293.15 K.

The number density of the gas can be computed using the Ideal Gas Law,

$$p = \frac{nRT}{N_{\mathrm{A}}}$$

where $R = 8.3144621$ J/(mol·K) is the universal gas constant and $N_{\mathrm{A}} = 6.02214129\cdot10^{23}$ 1/mol is the Avogadro constant. For a pressure of 1 Torr this yields $n = 3.294\cdot10^{22}$ atoms/m^3.

The interaction of ions with the background gas is modeled using the **Collisions** node with an **Elastic** subnode. At each time step taken by the solver, for each model particle a background gas particle is sampled at random from the Maxwellian distribution. The frequency of elastic collisions is then computed from the collision cross-section, background gas number density, and the relative velocity of the model particle with respect to the randomly sampled background gas molecule:

$$\nu = N_d\sigma|\mathbf{v}_p - \mathbf{v}_g|$$

where the collision cross section σ (SI unit: m^2) is usually a function of the particle kinetic energy. The collision probability is then computed as a function of the collision frequency and the time step size:

$$P = 1 - \exp(-\nu\Delta t)$$

The model uses three physics interfaces: Electrostatics, Electric Currents, and Charged Particle Tracing. The Electrostatics and Electric Currents interfaces are used to compute the DC and AC fields, respectively. These fields are then coupled to the Charged Particle Tracing interface, which models the motion of the ions due to the electric fields and interaction with neutral particles in the background gas. Interactions between the ions are neglected. In order to accurately model collisions of ions with the background gas, the average time between elastic collisions should be significantly greater than the maximum time step taken by the solver. Strict or manual time stepping is recommended.

Results and Discussion

The electric potential in the ion funnel is plotted in Figure 1. The gradual DC bias guides positive ions from the larger end of the funnel to the smaller end. The AC voltage, which is out of phase between adjacent electrodes, causes the gradient of the electric potential to become very large close to the electrodes, keeping the ions confined within the funnel.

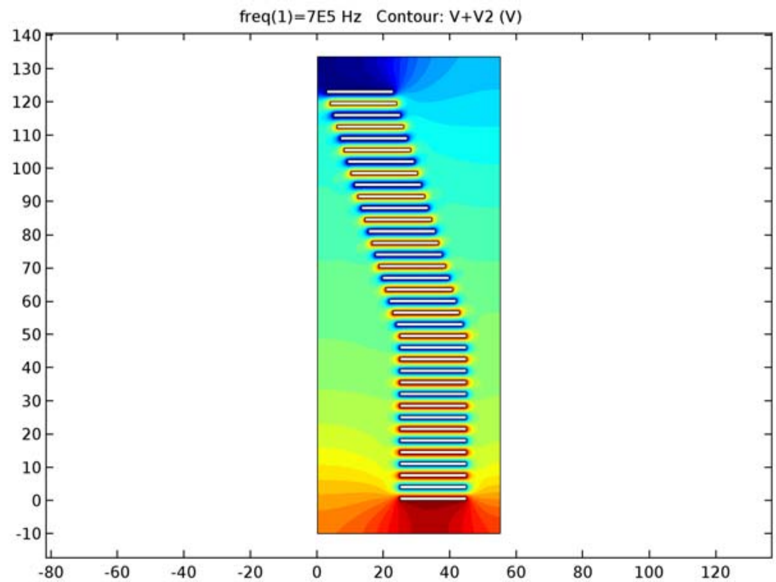


Figure 1: The combined DC and AC potential is plotted in the ion funnel at time t=0.

The ion trajectories are plotted in Figure 2. Because the ions are confined to an area of reduced size, they can be transported to another device, such as a mass spectrometer, more efficiently.

The x- and y-coordinates of the particles at the narrow end of the funnel are plotted in Figure 3. Although the ions are released along the positive x-axis, they are uniformly distributed around the z-axis by the time they reach the end of the funnel. Because the **Collisions** node uses random numbers to determine whether a collision takes place at each time step, the results may be slightly different from those shown in Figure 2 and Figure 3.

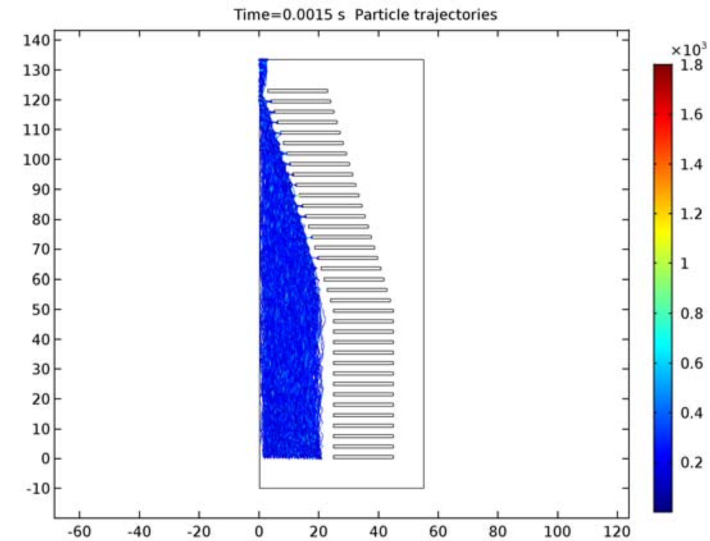


Figure 2: The trajectories of positive ions in the funnel.

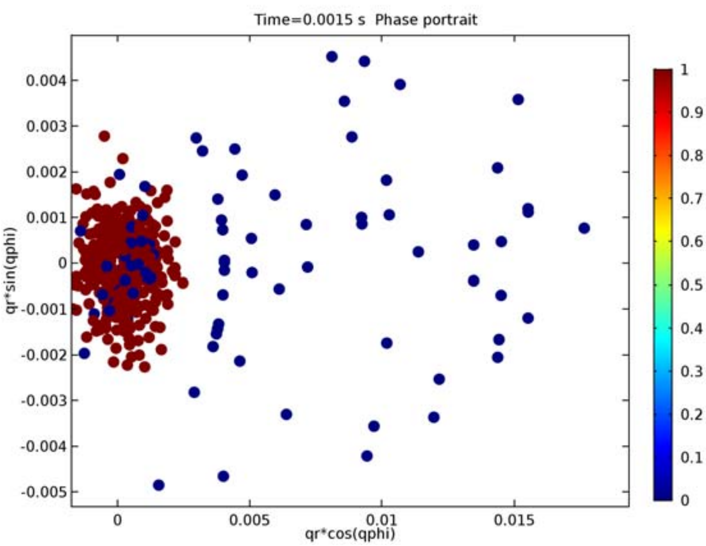


Figure 3: Phase portrait showing the x- and y-coordinates of the ions at the narrow end of the funnel. The color expression is red for particles that have exited the funnel and blue otherwise.

Reference

1. A. V. Phelps, "The application of scattering cross sections to ion flux models in discharge sheaths," *J. Appl. Phys.* vol. 76, pp. 747-753, 1994.

Application Library path: `ACDC_Module/Particle_Tracing/ion_funnel`

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 **2D Axisymmetric**.
2 In the **Select Physics** tree, select **AC/DC>Electrostatics (es)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **AC/DC>Electric Currents (ec)**.
5 Click **Add**.
6 In the **Select Physics** tree, select **AC/DC>Particle Tracing>Charged Particle Tracing (cpt)**.
7 Click **Add**.
8 Click **Study**.
9 In the **Select Study** tree, select **Custom Studies>Preset Studies for Some Physics Interfaces>Stationary**.
10 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

To save time, load the parameters from a file.

2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 Click **Load from File**.

4 Browse to the application's Application Libraries folder and double-click the file `ion_funnel_parameters.txt`.

GEOMETRY 1

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

2 In the **Settings** window for Geometry, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `welec`.

4 In the **Height** text field, type `telec`.

5 Locate the **Position** section. In the **r** text field, type `rmax`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `welec`.

4 In the **Height** text field, type `telec`.

5 Locate the **Position** section. In the **r** text field, type `rmax`.

6 In the **z** text field, type `telec+tgap`.

Create arrays containing the electrodes which are in phase with each other. This setup allows the number of electrodes to be changed at any time by changing the corresponding parameters.

Array 1 (arr1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **r1** only.

3 In the **Settings** window for Array, locate the **Size** section.

4 From the **Array type** list, choose **Linear**.

5 In the **Size** text field, type `(Nstraight+1)/2`.

6 Locate the **Displacement** section. In the **z** text field, type `2*(telec+tgap)`.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

8 From the **Show in physics** list, choose **Boundary selection**.

Array 2 (arr2)

1 Right-click **Array 1 (arr1)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

3 Select the object **r2** only.

4 In the **Settings** window for Array, locate the **Size** section.

5 From the **Array type** list, choose **Linear**.

6 In the **Size** text field, type `(Nstraight-1)/2`.

7 Locate the **Displacement** section. In the **z** text field, type `2*(telec+tgap)`.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

9 From the **Show in physics** list, choose **Boundary selection**.

Rectangle 3 (r3)

1 Right-click **Array 2 (arr2)** and choose **Build Selected**.

2 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

3 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

4 In the **Width** text field, type `welec`.

5 In the **Height** text field, type `telec`.

6 Locate the **Position** section. In the **r** text field, type `rmin`.

7 In the **z** text field, type `hfunnel-telec`.

Rectangle 4 (r4)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `welec`.

4 In the **Height** text field, type `telec`.

5 Locate the **Position** section. In the **r** text field, type `rmin+(rmax-rmin)/Ninclined`.

6 In the **z** text field, type `hfunnel-(2*telec+tgap)`.

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

Array 3 (arr3)

1 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **r3** only.

3 In the **Settings** window for Array, locate the **Size** section.

4 From the **Array type** list, choose **Linear**.

5 In the **Size** text field, type `(Ninclined+1)/2`.

6 Locate the **Displacement** section. In the **r** text field, type `2*(rmax-rmin)/Ninclined`.

7 In the **z** text field, type `-2*(telec+tgap)`.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

9 From the **Show in physics** list, choose **Boundary selection**.

Array 4 (arr4)

1 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **r4** only.

3 In the **Settings** window for Array, locate the **Size** section.

4 From the **Array type** list, choose **Linear**.

5 In the **Size** text field, type `(Ninclined-1)/2`.

6 Locate the **Displacement** section. In the **r** text field, type `2*(rmax-rmin)/Ninclined`.

7 In the **z** text field, type `-2*(telec+tgap)`.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

9 From the **Show in physics** list, choose **Boundary selection**.

Add a rectangle to enclose the modeling domain. Note that the rectangle is positioned a small distance away from the axis of symmetry. This makes the particle tracing model more robust, since the centrifugal force acting on the particles approaches infinity as the radial coordinate approaches zero.

Rectangle 5 (r5)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `rmax+welec+10[mm]`.

4 In the **Height** text field, type `hfunnel+20[mm]`.

5 Locate the **Position** section. In the **r** text field, type `0.2[mm]`.

6 In the **z** text field, type `-10[mm]`.

7 Click **Build All Objects**.

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

DEFINITIONS

Union 1

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

2 In the **Settings** window for Union, type `Odd electrodes` 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, In the **Selections to add** list, choose **Array 1** and **Array 4**.

6 Click **OK**.

Union 2

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

2 In the **Settings** window for Union, type `Even electrodes` 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, In the **Selections to add** list, choose **Array 2** and **Array 3**.

6 Click **OK**.

Union 3

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

2 In the **Settings** window for Union, type `All electrodes` 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, In the **Selections to add** list, choose **Odd electrodes** and **Even electrodes**.

6 Click **OK**.

Explicit 1

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

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

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

4 Select Boundaries 2, 3, and 144 only.

Union 4

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

2 In the **Settings** window for Union, type `All 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, In the **Selections to add** list, choose **All electrodes** and **Exterior boundaries**.

6 Click **OK**.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

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

2 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

ELECTROSTATICS (ES)

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

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

3 Click **Clear Selection**.

4 Select Domain 1 only.

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

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

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

4 Locate the **Electric Potential** section. In the V_0 text field, type `EDC*z`.

ELECTRIC CURRENTS (EC)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents (ec)**.

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

3 Click **Clear Selection**.

4 Select Domain 1 only.

Terminal 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Terminal**.

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

3 From the **Selection** list, choose **Odd electrodes**.

4 Locate the **Terminal** section. From the **Terminal type** list, choose **Voltage**.

5 In the V_0 text field, type `Vpp`.

Terminal 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Terminal**.

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

3 From the **Selection** list, choose **Even electrodes**.

4 Locate the **Terminal** section. From the **Terminal type** list, choose **Voltage**.

5 In the V_0 text field, type `-Vpp`.

CHARGED PARTICLE TRACING (CPT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Charged Particle Tracing (cpt)**.

2 In the **Settings** window for Charged Particle Tracing, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.

5 Locate the **Advanced Settings** section. Select the **Include out-of-plane degrees of freedom** check box.

Particle Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Charged Particle Tracing (cpt)** click **Particle Properties 1**.

2 In the **Settings** window for Particle Properties, locate the **Particle Mass** section.

3 In the m_p text field, type `0.146[kg/mol]/N_A_const`.

4 Locate the **Charge Number** section. In the Z text field, type `1`.

5 In the **Model Builder** window, click **Charged Particle Tracing (cpt)**.

Wall 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 Select Boundary 1 only.

3 In the **Settings** window for Wall, locate the **Wall Condition** section.

4 From the **Wall condition** list, choose **Bounce**.

Release from Grid 1

1 On the **Physics** toolbar, click **Global** and choose **Release from Grid**.

2 In the **Settings** window for Release from Grid, locate the **Initial Coordinates** section.

3 Click **r Range**.

4 In the **Range** dialog box, type 2 in the **Start** text field.

5 In the **Step** text field, type 1.

6 In the **Stop** text field, type 20.

7 Click **Replace**.

8 In the **Settings** window for Release from Grid, locate the **Initial Velocity** section.

9 From the **Initial velocity** list, choose **Maxwellian**.

10 In the $N_\mathbf{v}$ text field, type 5.

Electric Force 1

1 On the **Physics** toolbar, click **Domains** and choose **Electric Force**.

2 Select Domain 1 only.

3 In the **Settings** window for Electric Force, locate the **Electric Force** section.

4 From the $\mathbf{E}$ list, choose **Electric field (es/ccn1)**.

5 Locate the **Advanced Settings** section. Select the **Use piecewise polynomial recovery on field** check box.

Electric Force 2

1 On the **Physics** toolbar, click **Domains** and choose **Electric Force**.

2 Select Domain 1 only.

3 In the **Settings** window for Electric Force, locate the **Electric Force** section.

4 From the $\mathbf{E}$ list, choose **Electric field (ec/cucn1)**.

5 Locate the **Advanced Settings** section. Select the **Multiply force by phase angle** check box.

6 Select the **Use piecewise polynomial recovery on field** check box.

Now define the cross sections used for the collision model.

DEFINITIONS

Enter the analytic approximation for momentum cross section for elastic scattering between SF_6^+ ions and neutral Ar atoms from Ref. 1, which depends on the kinetic energy of the particles.

Analytic 1 (an1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for Analytic, type `Qm` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `1.15e-18*x^(-0.1)*(1+0.015/x)^0.6`.

4 Locate the **Units** section. In the **Arguments** text field, type `eV`.

5 In the **Function** text field, type `m^2`.

Enter the analytic approximation for isotropic elastic collision between SF_6^+ ions and neutral Ar atoms from Ref. 1, which depends on the kinetic energy of the particles.

Analytic 2 (an2)

1 On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for Analytic, type `Qi` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `2e-19/(x^(0.5)*(1+x))+3e-19*x/(1+x/3)^(2.3)`.

4 Locate the **Units** section. In the **Arguments** text field, type `eV`.

5 In the **Function** text field, type `m^2`.

CHARGED PARTICLE TRACING (CPT)

Collisions 1

1 On the **Physics** toolbar, click **Domains** and choose **Collisions**.

2 Select Domain 1 only.

3 In the **Settings** window for Collisions, locate the **Fluid Properties** section.

4 In the N_d text field, type `ND`.

5 Locate the **Collision Statistics** section. Select the **Count all collisions** check box.

Elastic 1

1 On the **Physics** toolbar, click **Attributes** and choose **Elastic**.

2 In the **Settings** window for Elastic, locate the **Collision Frequency** section.

3 In the σ text field, type `Qi(cpt.Ep)`.

4 Locate the **Collision Statistics** section. Select the **Count collisions** check box.

Collisions 1

In the **Model Builder** window, under **Component 1 (comp1)>Charged Particle Tracing (cpt)** click **Collisions 1**.

Resonant Charge Exchange 1

1. On the **Physics** toolbar, click **Attributes** and choose **Resonant Charge Exchange**.
2. In the **Settings** window for Resonant Charge Exchange, locate the **Collision Frequency** section.
3. In the σ text field, type `(Qm(cpt.Ep)-Qi(cpt.Ep))/2`.
4. Locate the **Collision Statistics** section. Select the **Count collisions** check box.

STUDY 1

Step 1: Stationary

1. In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2. In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.
3. In the table, clear the **Solve for** check box for **Electric Currents** and **Charged Particle Tracing**.

Frequency Domain

On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>Frequency Domain**.

Step 2: Frequency Domain

1. In the **Settings** window for Frequency Domain, locate the **Study Settings** section.
2. In the **Frequencies** text field, type `f0`.
3. Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Electrostatics** and **Charged Particle Tracing**.
4. Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
5. From the **Method** list, choose **Solution**.
6. From the **Study** list, choose **Study 1, Stationary**.

Step 1: Stationary

1. In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2. In the **Settings** window for Stationary, click to expand the **Values of dependent variables** section.
3. On the **Study** toolbar, click **Compute**.

RESULTS

Electric Potential (es)

Create a contour plot of the electric potential when $t = 0$.

2D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for 2D Plot Group, type `Sum of AC and DC Potentials` in the **Label** text field.

Contour 1

1 Right-click **Sum of AC and DC Potentials** and choose **Contour**.

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

3 In the **Expression** text field, type `V+V2`.

4 Locate the **Levels** section. In the **Total levels** text field, type `30`.

5 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

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

7 On the **Sum of AC and DC Potentials** toolbar, click **Plot**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar. Compare this plot to Figure 1. The plot indicates that the potential gradient is steep in the area surrounding the electrodes. The large electric field magnitude in this area confines ions within the funnel.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Time Dependent**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Electrostatics (es)** and **Electric Currents (ec)**.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

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

3 Click **Range**.

4 In the **Range** dialog box, type `1e-5` in the **Step** text field.

5 In the **Stop** text field, type `15e-4`.

6 Click **Replace**.

7 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

8 Select the **Relative tolerance** check box.

9 In the associated text field, type `1e-3`.

10 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

11 From the **Method** list, choose **Solution**.

12 From the **Study** list, choose **Study 1, Frequency Domain**.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for Time-Dependent Solver, click to expand the **Absolute tolerance** section.

4 Locate the **Absolute Tolerance** section. In the **Tolerance** text field, type `1e-4`.

5 Click to expand the **Time stepping** section. Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Strict**.

6 Select the **Maximum step** check box.

7 In the associated text field, type `1e-8`.

8 Click to expand the **Output** section. Clear the **Store reaction forces** check box.

9 Clear the **Store time derivatives** check box.

10 Clear the **Store solution on disk** check box.

11 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 3 (sol3)>Time-Dependent Solver 1** node, then click **Advanced**.

12 In the **Settings** window for Advanced, locate the **General** section.

13 From the **Solver log** list, choose **Minimal**.

14 On the **Study** toolbar, click **Compute**.

RESULTS

Particle Trajectories 1

1. In the **Model Builder** window, expand the **Particle Trajectories (cpt)** node, then click **Particle Trajectories 1**.
2. In the **Settings** window for Particle Trajectories, locate the **Coloring and Style** section.
3. Find the **Line style** subsection. From the **Type** list, choose **Line**.
4. On the **Particle Trajectories (cpt)** toolbar, click **Plot**. Compare the result to Figure 2.

2D Plot Group 7

1. On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
2. In the **Settings** window for 2D Plot Group, type `Transverse Particle Positions` in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Particle 1**.
4. From the **Time (s)** list, choose **0.0015**.
5. Locate the **Plot Settings** section. Clear the **Plot data set edges** check box.

Phase Portrait 1

1. On the **Transverse Particle Positions** toolbar, click **More Plots** and choose **Phase Portrait**.
2. In the **Settings** window for Phase Portrait, locate the **Expression** section.
3. From the **x-axis** list, choose **Manual**.
4. In the **Expression** text field, type `qr*cos(qphi)`.
5. From the **y-axis** list, choose **Manual**.
6. In the **Expression** text field, type `qr*sin(qphi)`.

Color Expression 1

1. Right-click **Phase Portrait 1** and choose **Color Expression**.
2. In the **Settings** window for Color Expression, locate the **Expression** section.
3. In the **Expression** text field, type `qz>120[mm]`.
4. On the **Transverse Particle Positions** toolbar, click **Plot**.
5. Click the **Zoom Extents** button on the **Graphics** toolbar. Compare the resulting plot to Figure 3.

Export

Create an animation showing the x- and y-components of the particle positions over time.

Animation 1

1 On the **Results** toolbar, click **Animation** and choose **File**.
2 In the **Settings** window for Animation, locate the **Target** section.
3 From the **Target** list, choose **Player**.
4 Locate the **Scene** section. From the **Subject** list, choose **Transverse Particle Positions**.
5 Right-click **Animation 1** and choose **Play**.

Finally, create a 1D plot showing the average radial position over time. Set the color expression to be the total radial force, averaged over all particles.

1D Plot Group 8

1 On the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for 1D Plot Group, type `Average Radial Particle Position` in the **Label** text field.
3 Locate the **Data** section. From the **Data set** list, choose **Particle 1**.

Particle 1

1 On the **Average Radial Particle Position** toolbar, click **Particle**.
2 In the **Settings** window for Particle, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Charged Particle Tracing>Position>qr - Particle position, r component**.
3 Locate the **Data Series Operation** section. From the **Operation** list, choose **Average**.

Color Expression 1

1 Right-click **Particle 1** and choose **Color Expression**.
2 In the **Settings** window for Color Expression, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Charged Particle Tracing>Forces>Total force>cpt.Ftr - Total force, r component**.
3 On the **Average Radial Particle Position** toolbar, click **Plot**.
4 Click the **Zoom Extents** button on the **Graphics** toolbar.

Created in COMSOL Multiphysics 5.2a

Iron Sphere in a 13.56 MHz Magnetic Field

Introduction

An iron sphere is exposed to a spatially uniform, sinusoidally time-varying, background magnetic field. The frequency of the field is so high that the skin depth in the sphere is much smaller than the radius. At such high frequencies it is possible to model only the fields and induced currents on the surface of the sphere, thus avoiding the need for solving for the fields within the volume of the sphere, resulting in significantly reduced model size.

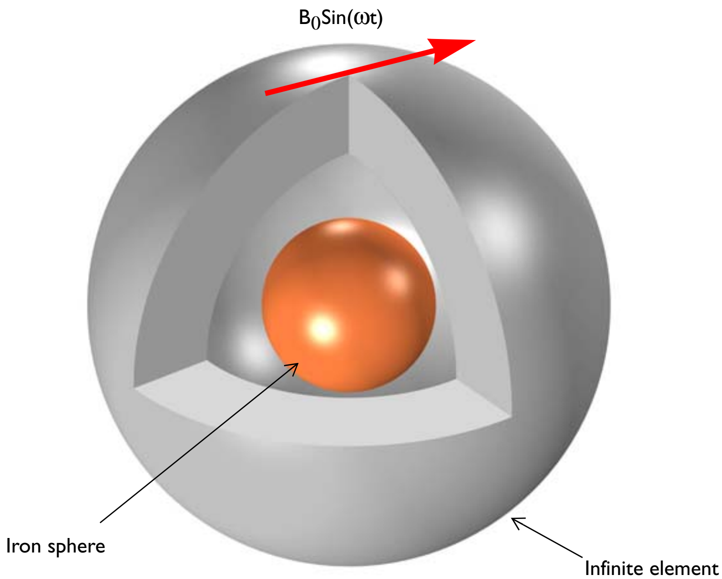


Figure 1: An iron sphere is exposed to a spatially uniform, sinusoidally time-varying, background magnetic field.

Model Definition

Figure 1 shows the setup, with an iron sphere placed in a spatially uniform, sinusoidally time-varying, background magnetic field applies using the Magnetic Fields, Reduced Field formulation. The model space is truncated by an Infinite Element Domain region. This is a domain condition that approximates a domain that extends to infinity. When using Infinite Element Domain features, the boundary condition on the outside of the modeling domain does not affect the solution.

At 13.56 MHz the skin depth of iron is ~0.65 µm. The surrounding air has $\varepsilon_r = 1$, $\mu_r = 1$, and $\sigma = 0$ S/m, which implies an infinite skin depth. Solve the model using the artificial conductivity approach by modifying the air conductivity to be $\sigma = 0.1$ S/m.

Because the skin depth in the iron sphere is much smaller than the sphere radius, it is possible to assume that the induced currents are only at the surface. This permits the use of the Impedance Boundary Condition on the iron sphere surface. The inside of the iron sphere is not modeled at all, since it is assumed that there are no significant currents, and negligible fields, within the sphere.

Results and Discussion

Figure 2 plots the magnetic field and the induced current density along with a visualization of the mesh at the sphere surface.

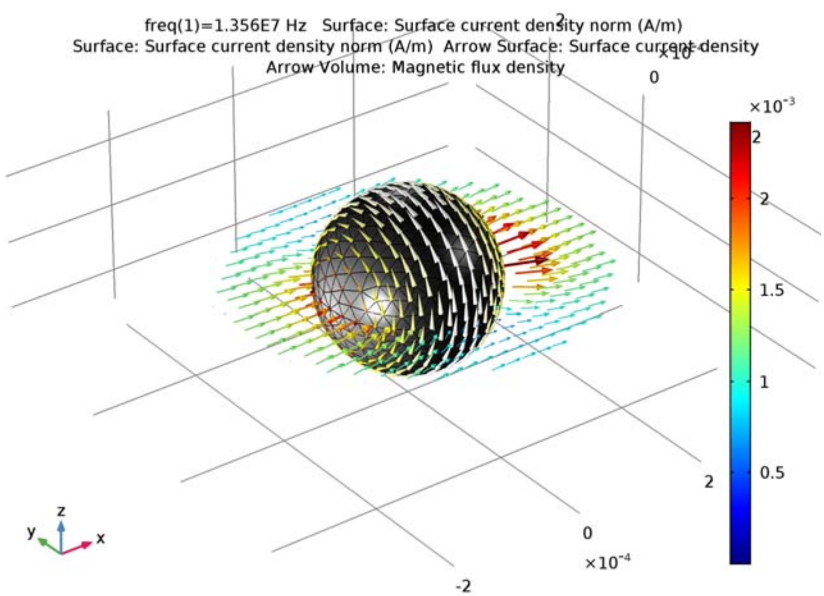


Figure 2: The induced currents on the surface of the iron sphere and the magnetic field in the surrounding space.

Application Library path: `ACDC_Module/Tutorials/iron_sphere_13mhz_bfield`

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>Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
B0	1[mT]	0.001 T	Background magnetic fields
r0	0.125[mm]	1.25E-4 m	Radius, iron sphere

GEOMETRY 1

Create a sphere with two layer definitions. The outermost layer represents the Infinite Element Domain and the core represents the iron sphere. The median layer is the air domain modified with a small amount of conductivity.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for Sphere, locate the **Size** section.

3 In the **Radius** text field, type `3*r0`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`r0`
`Layer 2`	`r0`

Suppress the iron sphere domain from the model domain.

Delete Entities 1 (del1)

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

2 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

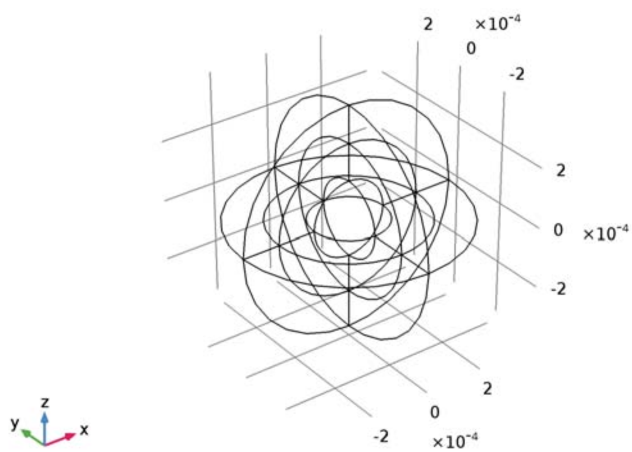
3 In the **Settings** window for Delete Entities, locate the **Entities or Objects to Delete** section.

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

5 On the object **sph1**, select Domain 9 only.

6 Click **Build All Objects**.

The surface of the iron sphere is now the exterior of the model domain and it is now possible to apply the impedance boundary condition.

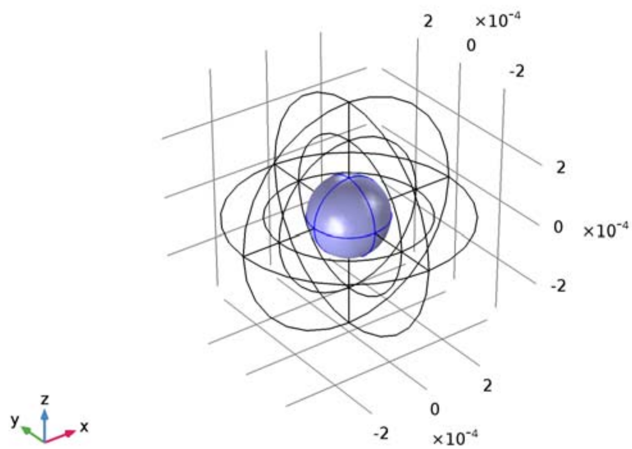


DEFINITIONS

Create a set of selections before setting up the physics. First, create a selection for the surface of the iron sphere.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for Explicit, locate the **Input Entities** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundaries 17–20, 31, 32, 39, and 42 only.



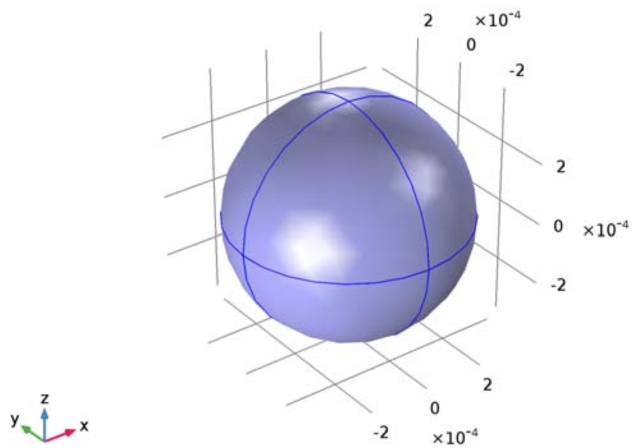
5 Right-click **Explicit 1** and choose **Rename**.
6 In the **Rename Explicit** dialog box, type `Iron surface` in the **New label** text field.
7 Click **OK**.

   Add an Infinite Element Domain on the outermost layer of the model domain.

Infinite Element Domain 1 (ie1)

1 On the **Definitions** toolbar, click **Infinite Element Domain**.

2 Select Domains 1–4, 9, 10, 13, and 16 only.



3 In the **Settings** window for Infinite Element Domain, locate the **Geometry** section.

4 From the **Type** list, choose **Spherical**.

MAGNETIC FIELDS (MF)

Set up the physics applying a uniform background magnetic fields. In the **Magnetic Fields** physics, the background field must be specified in terms of its vector potential field.

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

2 In the **Settings** window for Magnetic Fields, locate the **Background Field** section.

3 From the **Solve for** list, choose **Reduced field**.

4 Specify the $\mathbf{A}_b$ vector as

0	x
0	y
B0*y	z

Impedance Boundary Condition 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Impedance Boundary Condition**.

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

3 From the **Selection** list, choose **Iron surface**.

MATERIALS

Then, assign material properties. First, use air for all domains.

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

MATERIALS

Air (mat1)

Specify the conductivity of the air to a small value in order to improve the convergence rate.

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	`0.1[S/m]`	S/m	Basic

Override the core sphere surface with iron.

ADD MATERIAL

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

2 In the tree, select **Built-In>Iron**.

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

MATERIALS

Iron (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Iron (mat2)**.

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

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

4 From the **Selection** list, choose **Iron surface**.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MESH 1

Create an extra fine mesh on the surface of the iron sphere.

Size 1

1. In the **Model Builder** window, under **Component 1 (comp1)** 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 **Boundary**.
4. From the **Selection** list, choose **Iron surface**.
5. Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.

Free Tetrahedral 1

1. In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.
2. In the **Settings** window for Free Tetrahedral, click **Build All**.

   Plot the meshed structure to review the quality of the mesh.
3. On the **Mesh** toolbar, click **Plot**.

RESULTS

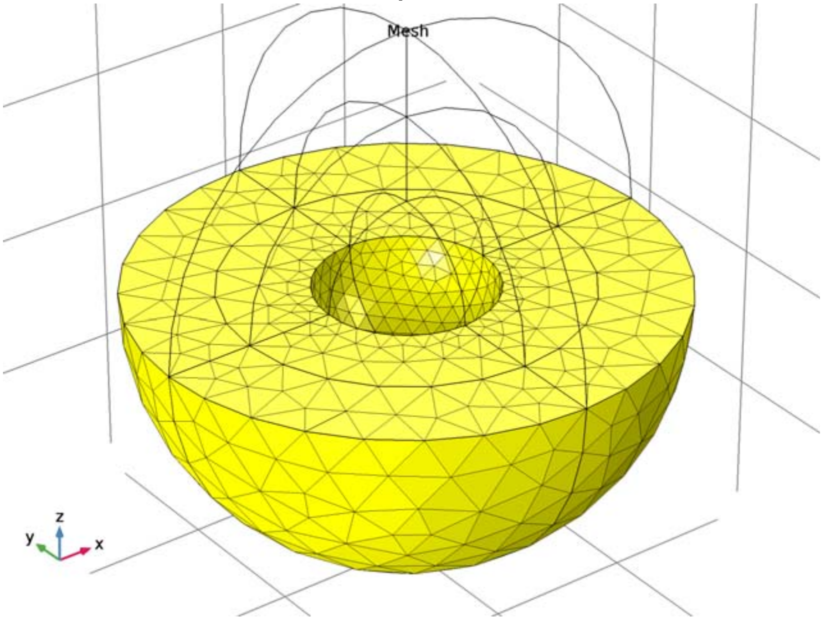
3D Plot Group 1

1. In the **Model Builder** window, under **Results** click **3D Plot Group 1**.
2. In the **Settings** window for 3D Plot Group, type `Mesh 1` in the **Label** text field.

Mesh 1

1. In the **Model Builder** window, under **Results>Mesh 1** click **Mesh 1**.
2. In the **Settings** window for Mesh, locate the **Level** section.
3. From the **Level** list, choose **Volume**.
4. Locate the **Color** section. From the **Element color** list, choose **Yellow**.
5. Click to expand the **Element filter** section. Locate the **Element Filter** section. Select the **Enable filter** check box.
6. In the **Expression** text field, type `z<0` to plot a section of the mesh.
7. On the **Mesh 1** toolbar, click **Plot**.

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



STUDY 1

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 `13.56[MHz]`.

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

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

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

7 On the **Home** toolbar, click **Compute**.

RESULTS

Study 1/Solution 1 (sol1)

Add a selection to the Data Set to plot quantities on the surface of the sphere.

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Solution 1 (sol1)**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 From the **Selection** list, choose **Iron surface**.

5 Select the **Propagate to lower dimensions** check box.

Create surface plots for the surface current density norm and the mesh on the sphere.

3D Plot Group 2

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for 3D Plot Group, type `Surface Current Density (mf)` in the **Label** text field.

Surface 1

1 Right-click **Surface Current Density (mf)** and choose **Surface**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields>Currents and charge>mf.normJs - Surface current density norm**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayPrint**.

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

5 Select the **Reverse color table** check box.

Surface 2

1 Right-click **Results>Surface Current Density (mf)>Surface 1** and choose **Duplicate**.

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

3 From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **Black**.

5 Select the **Wireframe** check box.

Surface Current Density (mf)

Add an arrow surface plot for the surface current density and an arrow volume plot showing the magnetic flux density.

Arrow Surface 1

1 In the **Model Builder** window, under **Results** right-click **Surface Current Density (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 **Model>Component 1>Magnetic Fields>Currents and charge>mf.Jsx,...,mf.Jsz - Surface current density**.

3 Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.

4 Select the **Scale factor** check box.

5 In the associated text field, type `6E-7`.

6 From the **Placement** list, choose **Mesh nodes**.

Color Expression 1

1 Right-click **Results>Surface Current Density (mf)>Arrow Surface 1** and choose **Color Expression**.

2 In the **Settings** window for Color Expression, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields>Currents and charge>mf.normJs - Surface current density norm**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

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

Surface Current Density (mf)

In the **Model Builder** window, under **Results** right-click **Surface Current Density (mf)** and choose **Arrow Volume**.

Arrow Volume 1

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

2 Find the **x grid points** subsection. In the **Points** text field, type `31`.

3 Find the **y grid points** subsection. In the **Points** text field, type `31`.

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

5 Right-click **Results>Surface Current Density (mf)>Arrow Volume 1** and choose **Color Expression**.

Color Expression 1

Compare the resulting plot with that in Figure 2.

Iron Sphere in a 20 kHz Magnetic Field

Introduction

An iron sphere is exposed to a spatially uniform, sinusoidally time-varying, background magnetic field. The frequency of the field is such that the skin depth is smaller than the sphere radius. The application computes the induced currents in the sphere and the perturbation to the background field. In addition, it addresses the question of how to properly mesh domains with significant skin effect.

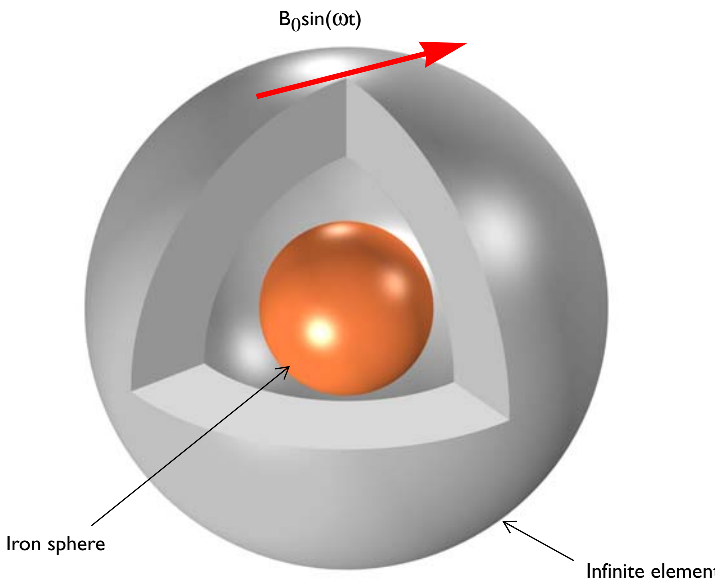


Figure 1: An iron sphere is exposed to a spatially uniform, sinusoidally time-varying, background magnetic field.

Model Definition

Figure 1 shows the setup, with an iron sphere placed in a spatially uniform, sinusoidally time-varying, background magnetic field applies using the Magnetic Fields, reduced field formulation. The model space is truncated by an Infinite Element Domain region. This is a domain condition that approximates a domain that extends to infinity. When using Infinite Element Domain features, the boundary condition on the outside of the modeling domain does not affect the solution.

The iron sphere has a relative permittivity of $\varepsilon_r = 1$, a relative permeability of $\mu_r = 4000$, and an electric conductivity of $\sigma = 1.12\times10^7$ S/m. The implicit assumption of modeling in the frequency domain is that all material properties are independent of the field strength. At the applied field strength of 1 mT, the permeability can be assumed to be constant—saturation effects in the iron are negligible.

At the operating frequency of 20 kHz, the skin depth in the iron is 16.8 µm. The surrounding air has $\varepsilon_r = 1$, $\mu_r = 1$, and $\sigma = 0$ S/m. This implies an infinite skin depth, which causes difficulties for the default solver. Instead, solve the model using the artificial conductivity approach by modifying the air conductivity to be $\sigma = 50$ S/m.

Due to the small skin depth, the solution can be assumed to have steep gradients normal to the boundary of the sphere. Such cases are well suited for boundary layer meshing, which inserts short triangular prismatic elements along the normal direction. The thickness of these prismatic elements should be equal to, or smaller than, the skin depth. In this way, the steep gradients normal to the boundary are better resolved by the mesh.

Results and Discussion

Figure 2 plots the magnetic field and the induced current density, while Figure 3 shows the mesh. The mesh resolves the skin effect well.

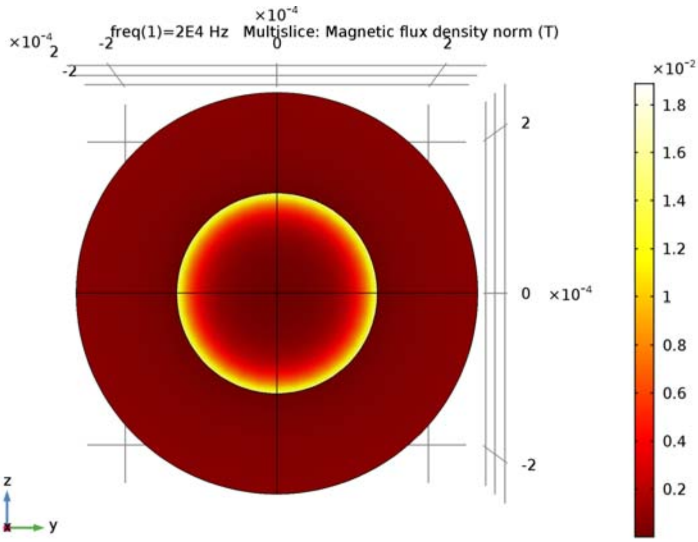


Figure 2: The induced currents and the magnetic field in the iron sphere.

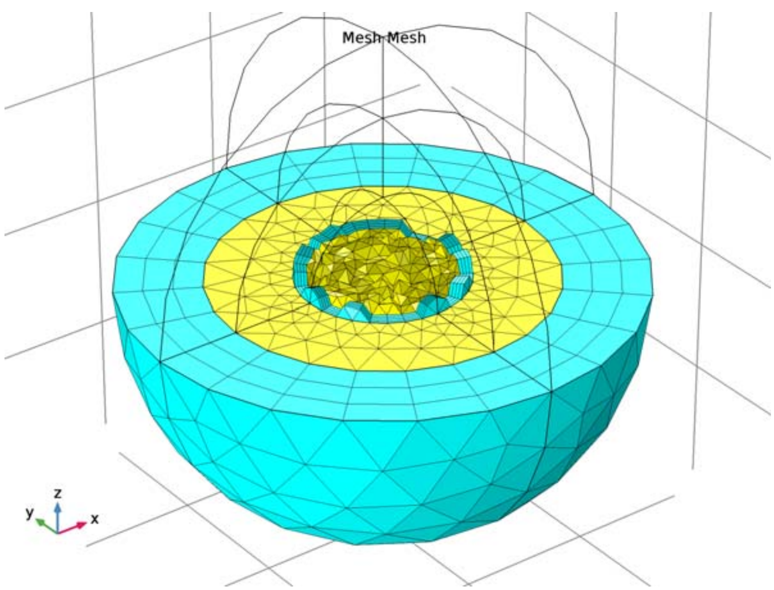


Figure 3: The mesh in the iron sphere and surrounding air.

Application Library path: ACDC_Module/Tutorials/iron_sphere_20khz_bfield

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>Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
B0	1[mT]	0.001 T	Background magnetic fields
r0	0.125[mm]	1.25E-4 m	Radius, iron sphere

GEOMETRY 1

Create a sphere with two layer definitions. The outermost layer represents the Infinite Element Domain and the core represents the iron sphere. The median layer is the air domain modified with a small amount of conductivity.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for Sphere, locate the **Size** section.

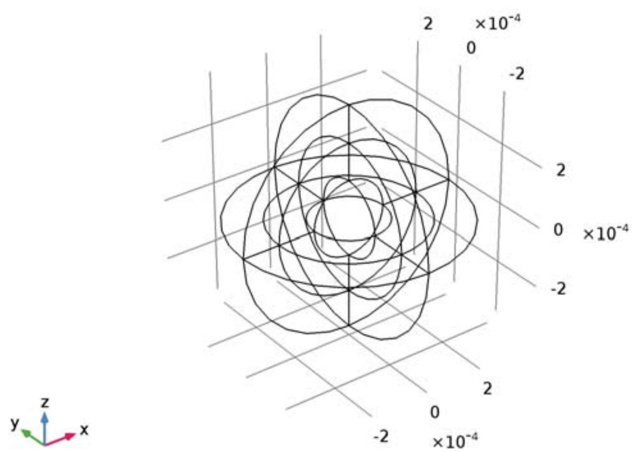
3 In the **Radius** text field, type `3*r0`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`r0`
`Layer 2`	`r0`

5 Click **Build All Objects**.

6 Click the **Wireframe Rendering** button on the **Graphics** toolbar.



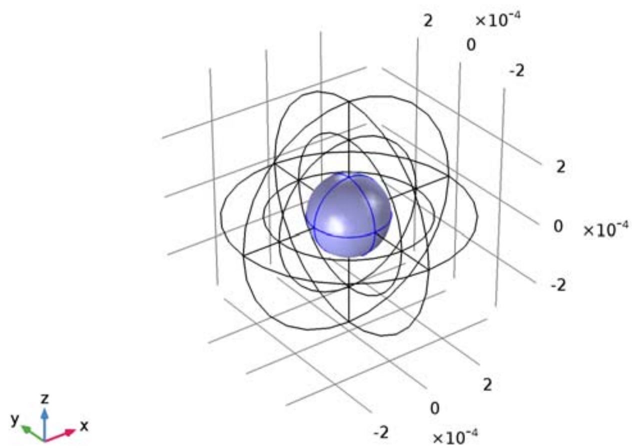
DEFINITIONS

Create a set of selections for use before setting up the physics. First, create a selection for the iron sphere.

Explicit 1

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

2 Select Domain 9 only.



3 Right-click **Explicit 1** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Core` in the **New label** text field.

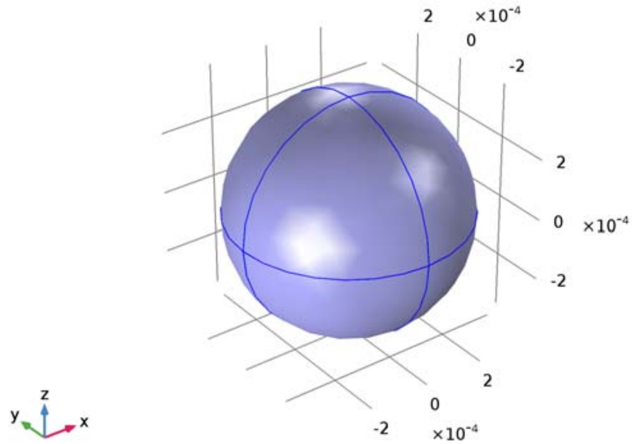
5 Click **OK**.

Add a selection for the Infinite Element domain.

Explicit 2

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

2 Select Domains 1–4, 10, 11, 14, and 17 only.



3 Right-click **Explicit 2** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Infinite Element domain` in the **New label** text field.

5 Click **OK**.

Add a selection for the domain in which to plot the magnetic flux density norm. It is the complement of the Infinite Element domains selection.

Complement 1

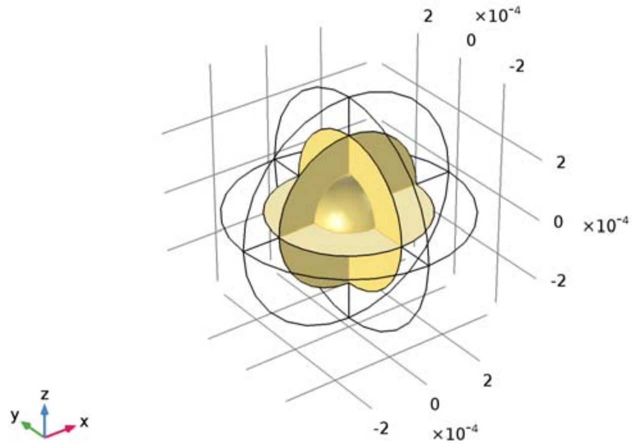
1 On the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for Complement, locate the **Input Entities** section.

3 Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **Infinite Element domain** in the **Selections to invert** list.

5 Click **OK**.



6 Right-click **Complement 1** and choose **Rename**.

7 In the **Rename Complement** dialog box, type `Analysis domain` in the **New label** text field.

8 Click **OK**.

Add an Infinite Element Domain. Use the selection **Infinite Element domains**.

Infinite Element Domain 1 (ie1)

1 On the **Definitions** toolbar, click **Infinite Element Domain**.

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

3 From the **Selection** list, choose **Infinite Element domain**.

4 Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

MAGNETIC FIELDS (MF)

Set up the physics applying a uniform background magnetic fields. In the **Magnetic Fields** physics, the background field must be specified in terms of its vector potential field.

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

2 In the **Settings** window for Magnetic Fields, locate the **Background Field** section.

3 From the **Solve for** list, choose **Reduced field**.

4 Specify the $\mathbf{A}_b$ vector as

0	x
0	y
B0*y	z

MATERIALS

Then, assign material properties. First, use air for all domains. Specify the conductivity of the air to a small value in order to improve the convergence rate.

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	50	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic

4 Right-click **Component 1 (comp1)>Materials>Material 1 (mat1)** and choose **Rename**.

5 In the **Rename Material** dialog box, type `Air` in the **New label** text field.

6 Click **OK**.

Override the core sphere with iron.

ADD MATERIAL

1 On 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>Iron**.

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

MATERIALS

Iron (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Iron (mat2)**.

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

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

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MESH 1

Create an extra fine mesh in the iron sphere, apply a boundary layer meshing at the surface of the iron sphere and use the swept mesh for the Infinite Element Domain.

Size 1

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

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.

Free Tetrahedral 1

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

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

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

4 From the **Selection** list, choose **Analysis domain**.

Boundary Layers 1

1 Right-click **Mesh 1** and choose **Boundary Layers**.

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

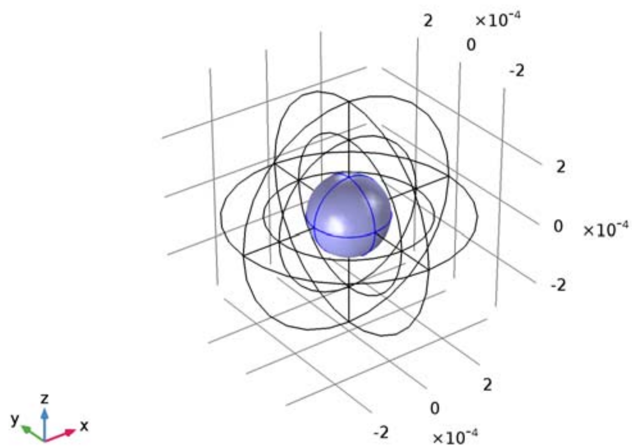
3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Core**.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

2 Select Boundaries 17–20, 31, 32, 39, and 42 only.



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

4 In the **Number of boundary layers** text field, type `4`.

5 In the **Boundary layer stretching factor** text field, type `1`.

6 From the **Thickness of first layer** list, choose **Manual**.

7 In the **Thickness** text field, type `8[um]`.

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

Swept 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Swept 1** and choose **Distribution**.

Distribution 1

1 In the **Settings** window for Distribution, locate the **Distribution** section.

2 In the **Number of elements** text field, type `3`.

3 Click **Build All**.

Plot the meshed structure to see the quality of the mesh as well as the boundary layer mesh on the iron sphere.

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

RESULTS

3D Plot Group 1

1 In the **Model Builder** window, under **Results** click **3D Plot Group 1**.
2 In the **Settings** window for 3D Plot Group, type `Mesh 1` in the **Label** text field.

Mesh 1

1 In the **Model Builder** window, under **Results>Mesh 1** click **Mesh 1**.
2 In the **Settings** window for Mesh, locate the **Level** section.
3 From the **Level** list, choose **Volume**.
4 From the **Element type** list, choose **Prism**.
5 Locate the **Color** section. From the **Element color** list, choose **Cyan**.
6 Click to expand the **Element filter** section. Locate the **Element Filter** section. Select the **Enable filter** check box.
7 In the **Expression** text field, type `z<0` to plot a section of the mesh.

Mesh 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mesh**.
2 In the **Settings** window for Mesh, locate the **Level** section.
3 From the **Level** list, choose **Volume**.
4 From the **Element type** list, choose **Tetrahedron**.
5 Locate the **Color** section. From the **Element color** list, choose **Yellow**.
6 Locate the **Element Filter** section. Select the **Enable filter** check box.
7 In the **Expression** text field, type `z<0`.
8 On the **Mesh 1** toolbar, click **Plot**.
9 Click the **Zoom In** button on the **Graphics** toolbar.

Compare the mesh with that shown in Figure 2.

STUDY 1

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 `20[kHz]`.
4 On the **Home** toolbar, click **Compute**.

Results

Magnetic Flux Density Norm (mf)

The default plot shows the magnetic flux density norm. Suppress the Infinite Element Domain for the result analysis. Add an arrow plot for the current density.

Multislice 1

1. In the **Model Builder** window, expand the **Magnetic Flux Density Norm (mf)** node, then click **Multislice 1**.
2. In the **Settings** window for Multislice, locate the **Coloring and Style** section.
3. From the **Color table** list, choose **ThermalLight**.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/Solution 1 (sol1)**.

Selection

1. On the **Results** toolbar, click **Selection**.
2. In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. From the **Selection** list, choose **Analysis domain**.
5. Select the **Propagate to lower dimensions** check box.

Magnetic Flux Density Norm (mf)

In the **Model Builder** window, under **Results** right-click **Magnetic Flux Density Norm (mf)** and choose **Arrow Volume**.

Arrow Volume 1

1. In the **Settings** window for Arrow Volume, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields>Currents and charge>mf.Jx,mf.Jy,mf.Jz - Current density**.
2. Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type 1.
3. Find the **y grid points** subsection. In the **Points** text field, type 31.
4. Find the **z grid points** subsection. In the **Points** text field, type 31.
5. Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.
6. Click the **Go to YZ View** button on the **Graphics** toolbar.

   Compare the reproduced plot with Figure 3.

Iron Sphere in a 60 Hz Magnetic Field

Introduction

An iron sphere is exposed to a spatially uniform, sinusoidally time-varying, background magnetic field. The frequency of the field is low enough that the skin depth is larger than the radius of the sphere. This application uses a reduced field formulation to impose the background field and demonstrates two approaches for solving the problem. The application computes the induced currents in the sphere and the perturbation to the background field.

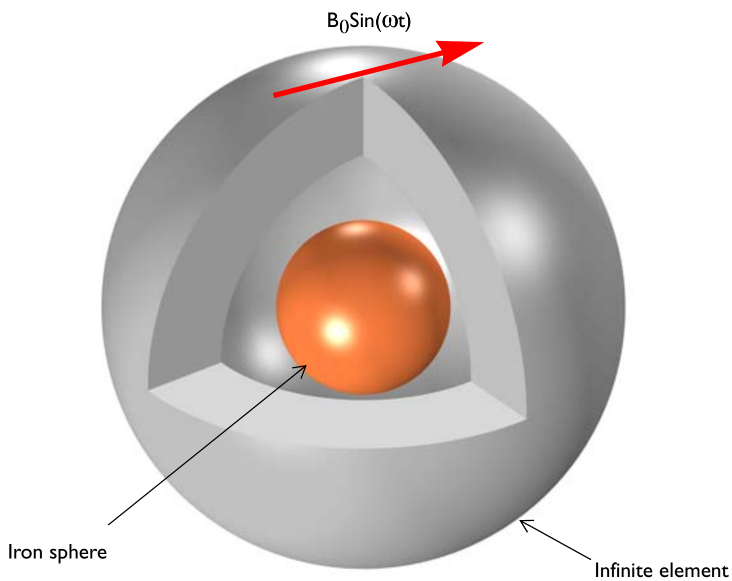


Figure 1: An iron sphere is exposed to a spatially uniform, sinusoidally time-varying, background magnetic field.

Model Definition

Figure 1 shows the setup, with an iron sphere placed in a spatially uniform, sinusoidally time-varying, background magnetic field applies using the Magnetic Fields, reduced field formulation. The model space is truncated by an Infinite Element Domain region. This is a domain condition that approximates a domain that extends to infinity. When using Infinite Element Domain features, the boundary condition on the outside of the modeling domain does not affect the solution.

The iron sphere has a relative permittivity of $\varepsilon_r = 1$, a relative permeability of $\mu_r = 4000$, and an electric conductivity of $\sigma = 1.12\times10^7$ S/m. The explicit assumption of modeling in the frequency domain is that all material properties are independent of the field strength. At the applied field strength of 1 mT, the permeability can be assumed to be constant—saturation effects in the iron are negligible.

For all models with time-varying magnetic fields, it is important to first consider the skin depth, δ, which is given by:

$$\delta = \frac{1}{Re\sqrt{i\omega\mu_0\mu_r(\sigma + i\omega\varepsilon_0\varepsilon_r)}}$$

At the operating frequency of 60 Hz the skin depth of iron is $\delta \sim 0.3$ mm. The surrounding air has $\varepsilon_r = 1$, $\mu_r = 1$, and $\sigma = 0$ S/m, thus the ratio of the largest to the smallest skin depth is infinite, and this leads to numerical difficulties when solving the problem.

It is possible to avoid this numerical difficulty by adding an artificial conductivity to the air domain. The basic concept behind this approach is to consider the skin depth in all the domains in the model and, in domains where the skin depth is very large or infinite, the conductivity should be increased. This artificial conductivity should be large enough so that the ratio of the largest to smallest skin depth be around 1000:1. The greater the artificial conductivity, the less accurate the results, but a too small artificial conductivity negatively affects converge.

An alternative approach, that does not require increasing the artificial conductivity as significantly, is to use gauge fixing. This adds an additional equation to the system of equations being solved, and as a consequence significantly increases the computational effort needed to solve the model.

Results and Discussion

Figure 2 plots the magnetic field and the induced current density for the model without gauge fixing, while Figure 3 shows the same results for the model with gauge fixing. The results agree well between the two approaches.

When using gauge fixing, the artificial conductivity of 5 S/m leads to a skin depth in the air of ~29 m and a total dissipation in the air domain of 4.53×10^{-11} nW. Without gauge fixing an artificial conductivity of 5000 S/m is used, leading to a skin depth in the air of ~0.9 m and a total dissipation in the air of 4.53×10^{-8} nW. For both approaches, the total dissipation in the sphere is 0.41×10^{-5} nW, two orders of magnitude higher.

Using gauge fixing increases the solution time and memory needed to solve the problem, and generally only slightly improves the solution. Therefore, it should be used sparingly. In any case, it is always recommended to carefully study the effects of artificial conductivity on the relative skin depths in the model, and keep in mind that this is a function of the operating frequency.

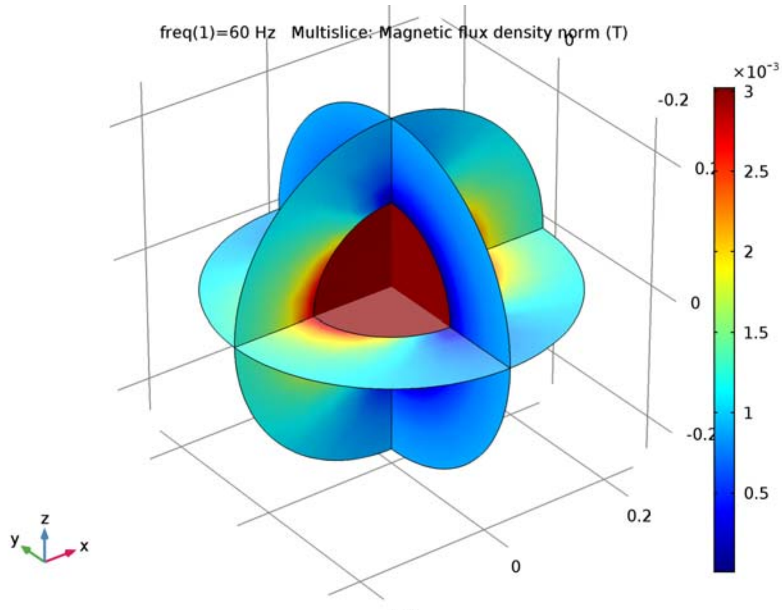


Figure 2: The induced currents and the magnetic field for the model without gauge fixing.

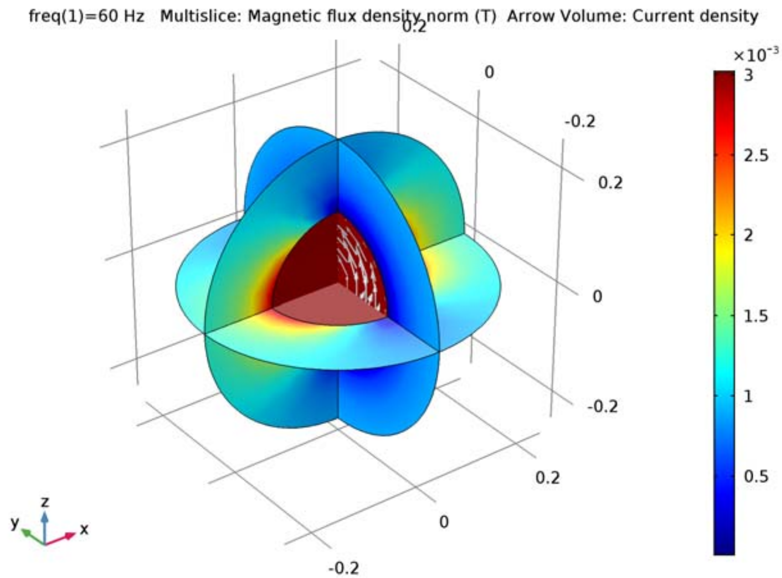


Figure 3: The induced currents and the magnetic field for the model with gauge fixing.

Application Library path: `ACDC_Module/Tutorials/iron_sphere_60hz_bfield`

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>Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`B0`	`1[mT]`	0.001 T	`Background magnetic fields`
`r0`	`0.125[mm]`	1.25E-4 m	`Radius, iron sphere`

GEOMETRY 1

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

2 In the **Settings** window for Geometry, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Create a sphere with two layers. The outermost layer represents the Infinite Element Domain and the core represents the iron sphere. The median layer is the air domain modified with a small amount of conductivity.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for Sphere, locate the **Size** section.

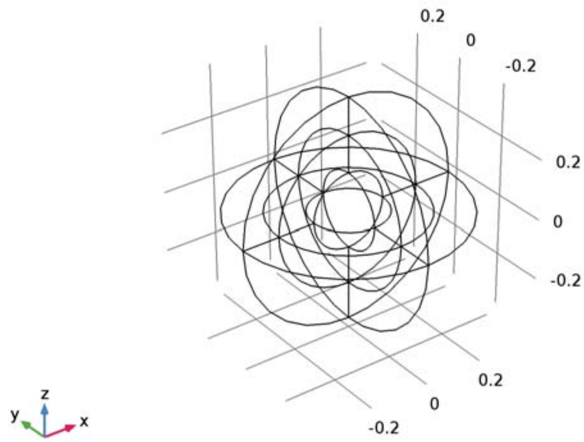
3 In the **Radius** text field, type `3*r0`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	`r0`
`Layer 2`	`r0`

5 Click **Build All Objects**.

6 Click the **Wireframe Rendering** button on the **Graphics** toolbar.



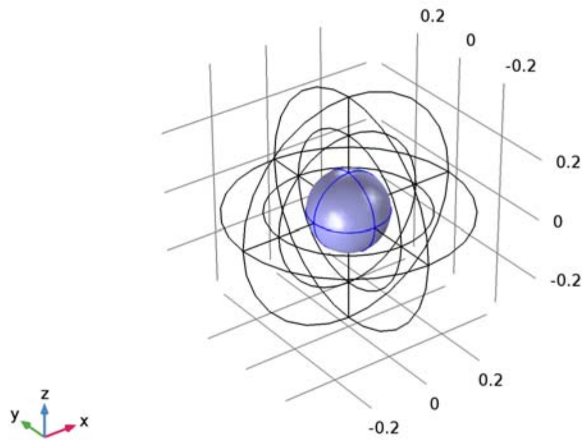
DEFINITIONS

Create a set of selections before setting up the physics. First, create a selection for the iron sphere.

Explicit 1

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

2 Select Domain 9 only.



3 Right-click **Explicit 1** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Core` in the **New label** text field.

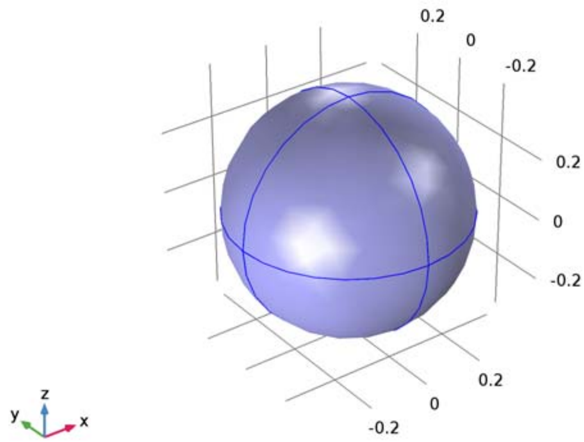
5 Click **OK**.

Add a selection for the Infinite Element Domain.

Explicit 2

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

2 Select Domains 1–4, 10, 11, 14, and 17 only.



3 Right-click **Explicit 2** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Infinite Element domains` in the **New label** text field.

5 Click **OK**.

Add a selection for the domain in which to plot the magnetic flux density norm. It is the complement of the Infinite Element domains selection.

Complement 1

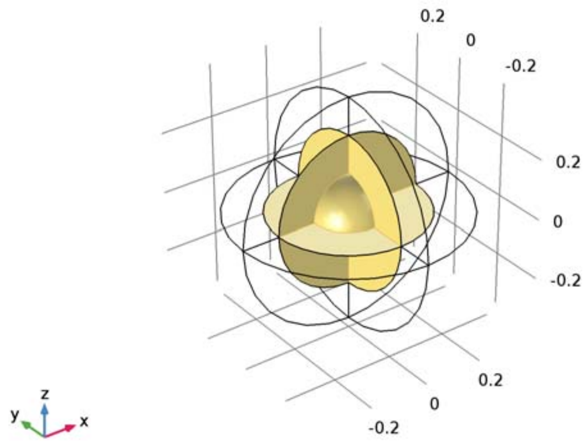
1 On the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for Complement, locate the **Input Entities** section.

3 Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **Infinite Element domains** in the **Selections to invert** list.

5 Click **OK**.



6 Right-click **Complement 1** and choose **Rename**.

7 In the **Rename Complement** dialog box, type `Analysis domain` in the **New label** text field.

8 Click **OK**.

Add an Infinite Element Domain. Use the selection Infinite Element domains.

Infinite Element Domain 1 (ie1)

1 On the **Definitions** toolbar, click **Infinite Element Domain**.

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

3 From the **Selection** list, choose **Infinite Element domains**.

4 Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

MAGNETIC FIELDS (MF)

Set up the physics applying a uniform background magnetic fields. In the **Magnetic Fields** physics, the background field must be specified in terms of its vector potential field.

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

2 In the **Settings** window for Magnetic Fields, locate the **Background Field** section.

3 From the **Solve for** list, choose **Reduced field**.

4 Specify the $\mathbf{A}_b$ vector as

0	x
0	y
`B0*y`	z

MATERIALS

Then, assign material properties. First, use air for all domains.

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Use an artificial conductivity of 5000 S/m.

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	5000	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic

4 Right-click **Component 1 (comp1)>Materials>Material 1 (mat1)** and choose **Rename**.

5 In the **Rename Material** dialog box, type `Air` in the **New label** text field.

6 Click **OK**.

Override the core sphere with iron.

ADD MATERIAL

1 On 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>Iron**.

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

MATERIALS

Iron (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Iron (mat2)**.

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

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

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MESH 1

Create a fine mesh for the iron sphere, and use a swept mesh for the Infinite Element Domain.

Size 1

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

5 Locate the **Element Size** section. From the **Predefined** list, choose **Finer**.

Free Tetrahedral 1

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

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

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

4 From the **Selection** list, choose **Analysis domain**.

5 Right-click **Mesh 1** and choose **Swept**.

Swept 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Swept 1** and choose **Distribution**.

Distribution 1

1 In the **Settings** window for Distribution, locate the **Distribution** section.

2 In the **Number of elements** text field, type 3.

3 Click **Build All**.

Plot the meshed structure to review the quality of the mesh.

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

RESULTS

3D Plot Group 1

1 In the **Model Builder** window, under **Results** click **3D Plot Group 1**.

2 In the **Settings** window for 3D Plot Group, type `Mesh 1` in the **Label** text field.

Mesh 1

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

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

3 From the **Level** list, choose **Volume**.

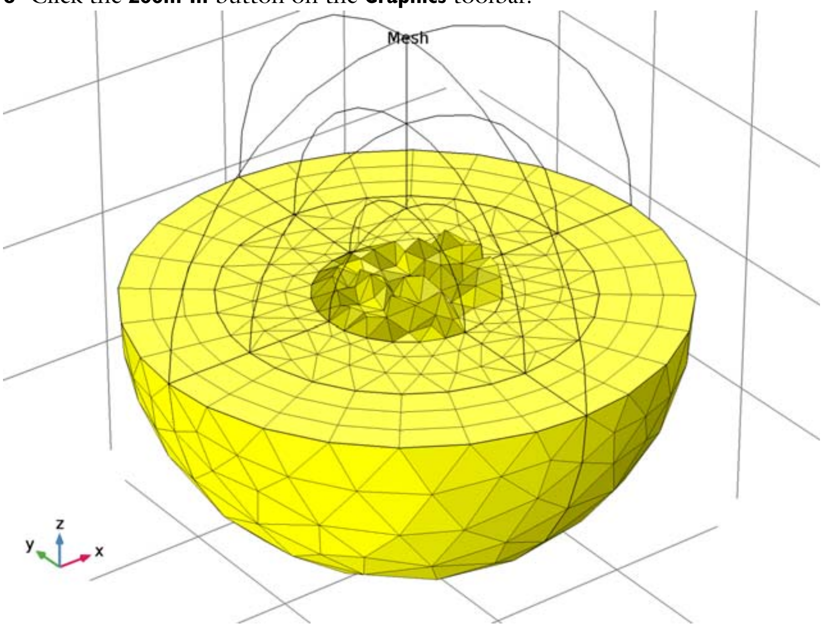
4 Locate the **Color** section. From the **Element color** list, choose **Yellow**.

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

6 In the **Expression** text field, type `z<0` to plot a section of the mesh.

7 On the **Mesh 1** toolbar, click **Plot**.

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



STUDY 1

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 `60[Hz]`.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density Norm (mf)

The default plot shows the magnetic flux density norm. Suppress the Infinite Element Domain for the result analysis and add an arrow plot for the current density.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/Solution 1 (sol1)**.

Selection

1. On the **Results** toolbar, click **Selection**.
2. In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. From the **Selection** list, choose **Analysis domain**.
5. Select the **Propagate to lower dimensions** check box.

Magnetic Flux Density Norm (mf)

In the **Model Builder** window, under **Results** right-click **Magnetic Flux Density Norm (mf)** and choose **Arrow Volume**.

Arrow Volume 1

1. In the **Settings** window for Arrow Volume, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields> Currents and charge>mf.Jx,mf.Jy,mf.Jz - Current density**.
2. Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `1`.
3. Find the **y grid points** subsection. In the **Points** text field, type `21`.
4. Find the **z grid points** subsection. In the **Points** text field, type `21`.
5. Locate the **Coloring and Style** section. From the **Color** list, choose **Gray**.

   Compare the reproduced plot with Figure 2.

   Add a plot for the skin depth.

3D Plot Group 3

1. On the **Results** toolbar, click **3D Plot Group**.
2. In the **Settings** window for 3D Plot Group, type `Skin depth (mf)` in the **Label** text field.

Slice 1

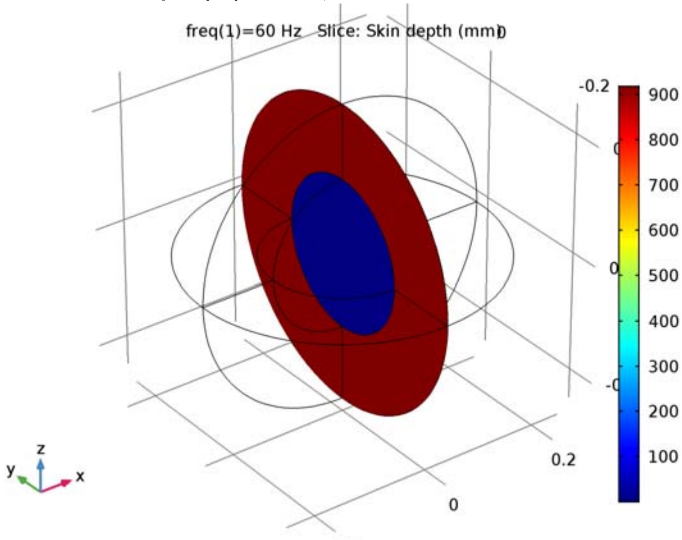
1 Right-click **Skin depth (mf)** and choose **Slice**.

2 In the **Settings** window for Slice, locate the **Plane Data** section.

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

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields>Material properties>mf.deltaS - Skin depth**.

5 On the **Skin depth (mf)** toolbar, click **Plot**.



The second study uses gauge fixing to improve convergence. The artificial conductivity in the air can therefore be reduced.

MATERIALS

Air (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Electrical conductivity	sigma	5	S/m	Basic

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

MAGNETIC FIELDS (MF)

Gauge Fixing for A-Field 1

1 On the **Physics** toolbar, click **Domains** and choose **Gauge Fixing for A-Field**.

2 In the **Settings** window for Gauge Fixing for A-Field, locate the **Domain Selection** section.

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

STUDY 2

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2** 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 `60[Hz]`.

The below adjustment on the Solver 2 settings is optional. You can skip this part and compute the study 2 without modifying the solver settings.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1>Iterative 1>Multigrid 1>Coarse Solver** node, then click **Direct**.

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

5 In the **Memory allocation factor** text field, type `2.5`.

The allocation factor is used to tell the direct linear solver MUMPS how much memory to allocate for the matrix factors. Sometimes the estimated memory is too low, in this case MUMPS increases the allocation factor and tries again. It may take a little shorter solution time by adjusting the allocation factor.

6 On the **Study** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density Norm (mf) 1

Specify a selection for the second solution, add an arrow and a skin depth plot as you did for the first study.

Study 2/Solution 2 (sol2)

In the **Model Builder** window, under **Results>Data Sets** click **Study 2/Solution 2 (sol2)**.

Selection

1 On the **Results** toolbar, click **Selection**.
2 In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **Analysis domain**.
5 Select the **Propagate to lower dimensions** check box.

Magnetic Flux Density Norm (mf) 1

In the **Model Builder** window, under **Results** right-click **Magnetic Flux Density Norm (mf) 1** and choose **Arrow Volume**.

Arrow Volume 1

1 In the **Settings** window for Arrow Volume, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields>Currents and charge>mf.Jx,mf.Jy,mf.Jz - Current density**.
2 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `1`.
3 Find the **y grid points** subsection. In the **Points** text field, type `21`.
4 Find the **z grid points** subsection. In the **Points** text field, type `21`.
5 Locate the **Coloring and Style** section. From the **Color** list, choose **Gray**.
6 On the **Magnetic Flux Density Norm (mf) 1** toolbar, click **Plot**.

   Compare the reproduced plot with Figure 3.

3D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for 3D Plot Group, type `Skin depth (mf) 1` in the **Label** text field.
3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

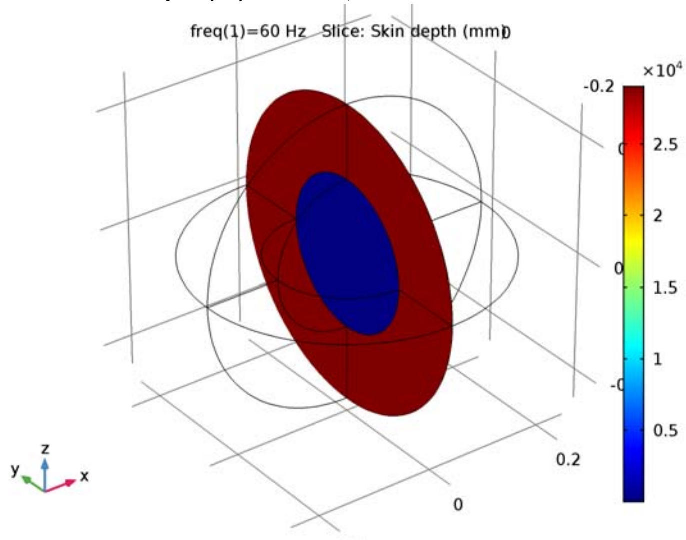
Slice 1

1 Right-click **Skin depth (mf) 1** and choose **Slice**.

2 In the **Settings** window for Slice, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields>Material properties>mf.deltaS - Skin depth**.

3 Locate the **Plane Data** section. In the **Planes** text field, type 1.

4 On the **Skin depth (mf) 1** toolbar, click **Plot**.



Derived Values

Finish the result analysis by evaluating the total dissipation, the volume integration of the resistive losses for each air and iron domain.

Volume Integration 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Volume Integration**.

2 Select Domains 5–8, 12, 13, 15, and 16 only.

3 In the **Settings** window for Volume Integration, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Magnetic Fields>Heating and losses>mf.Qrh - Volumetric loss density, electric**.

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

Expression	Unit	Description
mf.Qrh	nW	Volumetric loss density, electric

5 Click **Evaluate**.

Volume Integration 2

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Volume Integration**.

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

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 Select Domains 5–8, 12, 13, 15, and 16 only.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Magnetic Fields>Heating and losses>mf.Qrh - Volumetric loss density, electric**.

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

Expression	Unit	Description
mf.Qrh	nW	Volumetric loss density, electric

7 Click **Table 1 - Volume Integration 1 (mf.Qrh)**.

Volume Integration 3

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Volume Integration**.

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

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

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Magnetic Fields>Heating and losses>mf.Qrh - Volumetric loss density, electric**.

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

Expression	Unit	Description
mf.Qrh	nW	Volumetric loss density, electric

6 Click **Table 1 - Volume Integration 1 (mf.Qrh)**.

Volume Integration 4

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Volume Integration**.
2 In the **Settings** window for Volume Integration, locate the **Data** section.
3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.
4 Locate the **Selection** section. From the **Selection** list, choose **Core**.
5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Magnetic Fields>Heating and losses>mf.Qrh - Volumetric loss density, electric**.
6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
mf.Qrh	nW	Volumetric loss density, electric

7 Click **Table 1 - Volume Integration 1 (mf.Qrh)**.

Magnetic Lens

Introduction

Scanning electron microscopes take images of samples by scanning with a high-energy beam of electrons. The subsequent electron interactions produce signals such as secondary and back-scattered electrons that contain information about the sample surface topography. Electromagnetic lenses are used to focus this electron beam down to a spot about 10 nm wide on the sample surface.

Note: This application requires the AC/DC Module and the Particle Tracing Module.

Model Definition

Particles (electrons) are released from near the bottom boundary of the simulation space and pass through a collimator. This collimator can typically be adjusted to remove stray electrons. A simple DC coil produces an axial magnetic field. This rotationally symmetric, inhomogeneous magnetic field results in non-axial electrons experiencing a radial force causing them to spiral about the axis. As they begin to spiral, they have a larger velocity component perpendicular to the mainly axial magnetic field, therefore the radius of their spiral/helical path decreases. Thus, a parallel beam of electrons entering the lens converges to a point.

If the region in which the magnetic field acts upon the electrons is sufficiently small, this coil acts as a 'thin' convex lens and the thin lens expression holds.

MODEL EQUATIONS

A simple model is set up to test the magnetic force within the Charged Particle Tracing interface. The equations solved are the equation of motion of a charged particle in a magnetic field (Lorentz force):

$$\frac{d}{dt}(m\mathbf{v}) = q(\mathbf{v}\times\mathbf{B})$$

where q (SI unit: C) is the particle charge, $\mathbf{v}$ (SI unit: m/s) is the particle velocity, and $\mathbf{B}$ (SI unit: T) is the magnetic flux density. The total work done on a particle by a magnetic field is zero.

Results and Discussion

The magnetic flux density is plotted in Figure 1. The strength of the lens depends upon the coil configuration and current. The lenses within electron microscopes are generally very strong, in some cases focusing the electron beam within the lens itself.

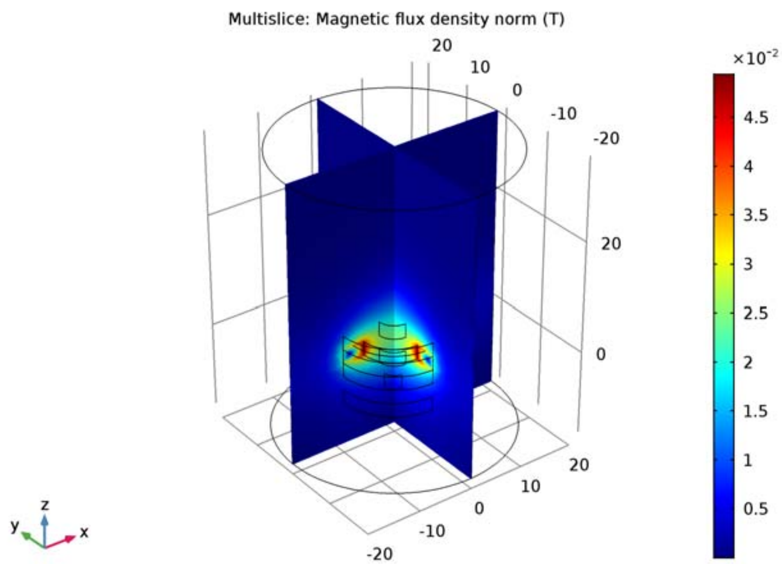


Figure 1: Plot of the magnetic flux density in the magnetic lens.

Figure 2 plots the electron trajectories as they travel through the coil. The electrons are focused at a point along the z-axis. The focal length is given by:

$$f = K\frac{V}{i^2}$$

where K is a constant based on the coil geometry and number of turns, V is the accelerating voltage and i is the coil current. The focal length increases with electron energy (that is, V) because their high velocity means they spend less time experiencing a force due the magnetic field. However, as the current increases so does the magnetic field strength, therefore the electrons spiral in tighter paths bringing the focal length closer.

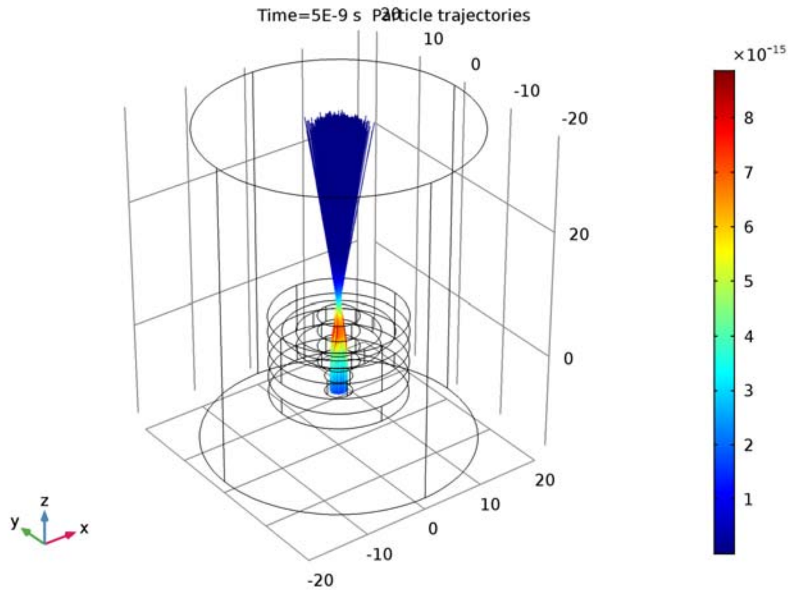


Figure 2: Plot of the electron trajectories traveling through the magnetic lens.

The ability to change the focal length of a lens is useful as it allows the focusing onto a surface in addition to adjusting the magnification. The effect of the focusing can be seen in Figure 3 which shows a Poincaré map of the particle position at three different snapshots in time. The sharpness of the cross-over can be improved using multiple lenses.

When charged particle beams are released, additional global variables are used to define beam properties such as the emittance and the Twiss parameters. These global variables can be used to characterize the shape of a beam and the transverse phase space distribution of the beam particles. In Figure 4 the hyperemittance is plotted along the average beam trajectory as a color expression and as a tube radius expression. The nominal trajectory reaches maximum thickness shortly after entering the lens and appears to be pinched off at the location where the beam is focused.

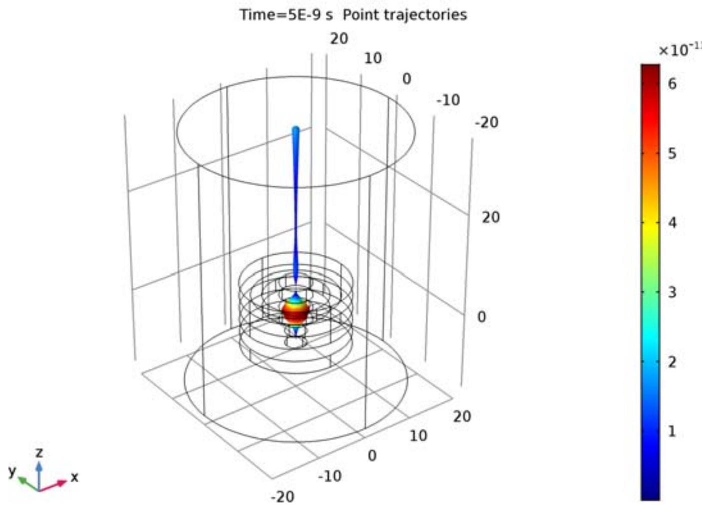


Figure 3: The nominal beam trajectory is plotted, with a color and thickness proportional to the 1-rms hyperemittance of the beam...

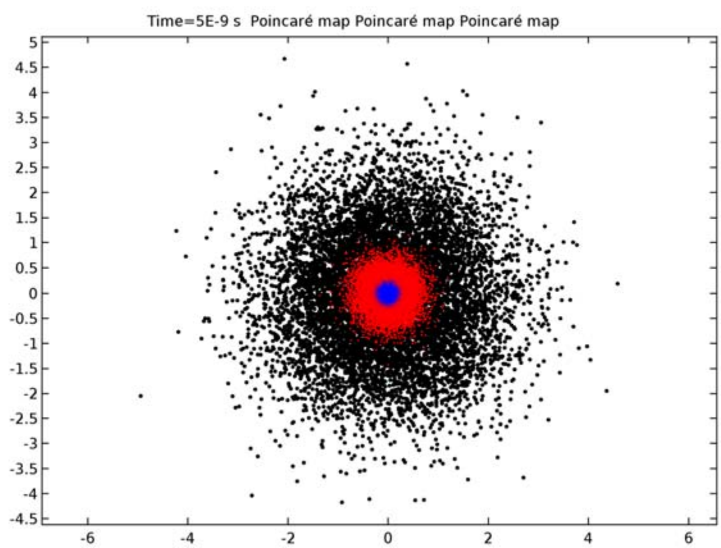


Figure 4: Poincaré maps of the particle location in the xy-plane initially (red), at the focal point of the lens (blue), and at the last time step (black).

Reference

1. M.J. Pritchard, *Manipulation of Ultracold Atoms Using Magnetic and Optical Fields*, PhD thesis, Durham University, September 2006, http://massey.dur.ac.uk/resources/mjpritchard/thesis_pritchard.pdf.

Application Library path: `ACDC_Module/Particle_Tracing/magnetic_lens`

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>Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
Ic	0.32[A]	0.32 A	Coil current
Nc	1000	1000	Number of turns in coil

GEOMETRY 1

The coil geometry is constructed using cylinders, and it is available as a separate file in the Application Library. Insert the prepared geometry sequence from the file. You can read the instructions for creating the geometry in the appendix.

1 On the **Geometry** toolbar, click **Insert Sequence**.
2 Browse to the application's Application Libraries folder and double-click the file `magnetic_lens_geom_sequence.mph`.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

1 In the **Settings** window for Material, locate the **Material Contents** section.
2 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	6e7	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.
2 Select Domain 1 only.
3 In the **Settings** window for Material, locate the **Material Contents** section.
4 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Relative permeability	mur	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic

MAGNETIC FIELDS (MF)

Coil 1

1 On the **Physics** toolbar, click **Domains** and choose **Coil**.

2 Select Domain 4 only.

3 In the **Settings** window for Coil, locate the **Coil** section.

4 From the **Conductor model** list, choose **Homogenized multi-turn**.

5 From the **Coil type** list, choose **Circular**.

6 Locate the **Homogenized Multi-Turn Conductor** section. In the N text field, type `Nc`.

7 Locate the **Coil** section. In the I_{coil} text field, type `Ic`.

Specify the reference edges to be used in the calculation of the current path for the circular coil. To obtain the best results, the selected edges should have a radius close to the average coil radius. In this case, select the edges created for this purpose in previous steps.

Coil Geometry 1

1 In the **Model Builder** window, expand the **Coil 1** node, then click **Coil Geometry 1**.

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

3 Click **Clear Selection**.

4 Select Edges 22, 23, 57, and 82 only.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Scale**.

Scale 1

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

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

3 Select Domains 2–5 only.

4 Locate the **Scale** section. In the **Element size scale** text field, type `0.5`.

Free Triangular 1

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

2 Select Boundary 30 only.

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

Use a fine mesh on the surface where particles will be released.

Size 1

1 In the **Settings** window for Size, locate the **Element Size** section.

2 From the **Predefined** list, choose **Extremely fine**.

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

Free Tetrahedral 1

In the **Settings** window for Free Tetrahedral, click **Build All**.

STUDY 1

1 On the **Home** toolbar, click **Compute**.

2 In the **Model Builder** window, expand the **Study 1** node.

RESULTS

Multislice 1

1 In the **Model Builder** window, expand the **Results>Magnetic Flux Density Norm (mf)** node, then click **Multislice 1**.

2 In the **Settings** window for Multislice, locate the **Multiplane Data** section.

3 Find the **z-planes** subsection. In the **Planes** text field, type `0`.

4 On the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**. Compare the resulting image to Figure 1.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **AC/DC>Particle Tracing>Charged Particle Tracing (cpt)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

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

6 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Time Dependent**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Magnetic Fields (mf)** interface.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

CHARGED PARTICLE TRACING (CPT)

On the **Physics** toolbar, click **Magnetic Fields (mf)** and choose **Charged Particle Tracing (cpt)**.

GEOMETRY 1

In the **Model Builder** window, collapse the **Component 1 (comp1)>Geometry 1** node.

DEFINITIONS

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

CHARGED PARTICLE TRACING (CPT)

1. In the **Settings** window for Charged Particle Tracing, locate the **Domain Selection** section.
2. Click **Clear Selection**.
3. Select Domain 1 only.

   You need to provide the forces acting on the particles; in this case, the magnetic (Lorentz) force.

Magnetic Force 1

1. On the **Physics** toolbar, click **Domains** and choose **Magnetic Force**.
2. Select Domain 1 only.
3. In the **Settings** window for Magnetic Force, locate the **Magnetic Force** section.
4. From the $\mathbf{B}$ list, choose **Magnetic flux density (mf)**.

Particle Beam 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Particle Beam**.
2. Select Boundary 30 only.
3. In the **Settings** window for Particle Beam, locate the **Initial Position** section.
4. In the N text field, type `10000`.
5. Locate the **Initial Transverse Velocity** section. From the **Sampling from phase space ellipse** list, choose **Gaussian**.
6. In the $\varepsilon_{\mathrm{rms}}$ text field, type `0.1[um]`.
7. Locate the **Initial Longitudinal Velocity** section. In the E text field, type `0.5[keV]`.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

2 In the **Settings** window for Time Dependent, click to expand the **Values of dependent variables** section.

3 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Study 1, Stationary**.

6 Locate the **Study Settings** section. Click **Range**.

7 In the **Range** dialog box, choose **Number of values** from the Entry method list.

8 In the **Stop** text field, type `5e-9`.

9 In the **Number of values** text field, type `50`.

10 Click **Replace**.

11 On the **Home** toolbar, click **Compute**.

RESULTS

Particle Trajectories (cpt)

In the **Model Builder** window, expand the **Particle Trajectories (cpt)** node.

Particle Trajectories 1

1 In the **Model Builder** window, expand the **Results>Particle Trajectories (cpt)>Particle Trajectories 1** node, then click **Particle Trajectories 1**.

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

3 Find the **Line style** subsection. From the **Type** list, choose **Line**.

4 Find the **Point style** subsection. From the **Type** list, choose **None**.

Color Expression 1

1 In the **Model Builder** window, under **Results>Particle Trajectories (cpt)>Particle Trajectories 1** click **Color Expression 1**.

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

3 In the **Expression** text field, type `sqrt(cpt.Ftx^2+cpt.Fty^2+cpt.Ftz^2)`.

4 On the **Particle Trajectories (cpt)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar. Compare the resulting image to Figure 2.

Now observe the beam hyperemittance along the nominal beam trajectory.

Average Beam Position (cpt)

1 In the **Model Builder** window, under **Results** click **Average Beam Position (cpt)**.

2 In the **Settings** window for 3D Plot Group, type `Average Beam Position and Hyperemittance` in the **Label** text field.

Point Trajectories 1

1 In the **Model Builder** window, expand the **Results>Average Beam Position and Hyperemittance** node, then click **Point Trajectories 1**.

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

3 Find the **Line style** subsection. From the **Type** list, choose **Tube**.

4 Click **Replace Expression**.

5 Right-click and choose **Charged Particle Tracing>Beam properties>cpt.e1hrms - 1-RMS beam hyperemittance** from the menu.

6 Select the **Radius scale factor** check box.

7 In the associated text field, type `4E10`.

8 From the **Interpolation** list, choose **Uniform**.

9 On the **Average Beam Position and Hyperemittance** toolbar, click **Plot**. Compare the resulting image to Figure 3.

Now construct a **Poincaré Map** to visualize the radial distribution of particles initially, at the focal point, and at the exit of the modeling domain.

Cut Plane 1

1 On the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for Cut Plane, locate the **Plane Data** section.

3 From the **Plane** list, choose **xy-planes**.

4 In the **z-coordinate** text field, type `-6`.

5 Locate the **Data** section. From the **Data set** list, choose **Particle 1**.

Cut Plane 2

1 Right-click **Cut Plane 1** and choose **Duplicate**.

2 In the **Settings** window for Cut Plane, locate the **Plane Data** section.

3 In the **z-coordinate** text field, type `7`.

Cut Plane 3

1 Right-click **Results>Data Sets>Cut Plane 2** and choose **Duplicate**.

2 In the **Settings** window for Cut Plane, locate the **Plane Data** section.

3 In the **z-coordinate** text field, type `34`.

2D Plot Group 4

1 On the **Results** toolbar, click **2D Plot Group**.

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

Poincaré Map 1

1 On the **Poincaré Maps** toolbar, click **More Plots** and choose **Poincaré Map**.

2 In the **Settings** window for Poincaré Map, locate the **Data** section.

3 From the **Cut plane** list, choose **Cut Plane 3**.

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

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

6 Select the **Radius scale factor** check box.

7 In the associated text field, type `4E-2`.

Poincaré Map 2

1 Right-click **Poincaré Map 1** and choose **Duplicate**.

2 In the **Settings** window for Poincaré Map, locate the **Data** section.

3 From the **Cut plane** list, choose **Cut Plane 1**.

4 Locate the **Coloring and Style** section. In the **Radius scale factor** text field, type `2E-2`.

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

Poincaré Map 3

1 Right-click **Results>Poincaré Maps>Poincaré Map 2** and choose **Duplicate**.

2 In the **Settings** window for Poincaré Map, locate the **Data** section.

3 From the **Cut plane** list, choose **Cut Plane 2**.

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

5 In the **Radius scale factor** text field, type `5E-3`.

6 On the **Poincaré Maps** toolbar, click **Plot**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar. Compare the resulting image to Figure 4.

On the **Home** toolbar, click **Add Component** and choose **3D**.

GEOMETRY 1

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

2 In the **Settings** window for Geometry, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `10`.

4 In the **Height** text field, type `2.5`.

Cylinder 2 (cyl2)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `6`.

4 In the **Height** text field, type `2.5`.

5 Right-click **Cylinder 2 (cyl2)** and choose **Build Selected**.

Cylinder 1 (cyl1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.

Cylinder 3 (cyl3)

1 In the **Settings** window for Cylinder, locate the **Position** section.

2 In the **z** text field, type `-7.5`.

Cylinder 4 (cyl4)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `2`.

4 In the **Height** text field, type `2.5`.

5 Locate the **Position** section. In the **z** text field, type `-7.5`.

Cylinder 1 (cyl1)

Right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.

Cylinder 5 (cyl5)

1 In the **Settings** window for Cylinder, locate the **Position** section.

2 In the **z** text field, type `-2.5`.

Cylinder 6 (cyl6)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `3`.

4 In the **Height** text field, type `2.5`.

5 Locate the **Position** section. In the **z** text field, type `-2.5`.

Cylinder 1 (cyl1)

Right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.

Cylinder 7 (cyl7)

1 In the **Settings** window for Cylinder, locate the **Position** section.

2 In the **z** text field, type `2.5`.

Cylinder 8 (cyl8)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `3`.

4 In the **Height** text field, type `2.5`.

5 Locate the **Position** section. In the **z** text field, type `2.5`.

Cylinder 9 (cyl9)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.

3 In the **Radius** text field, type `20`.

4 In the **Height** text field, type `50`.

5 Locate the **Position** section. In the **z** text field, type `-15`.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the objects **cyl5**, **cyl3**, **cyl1**, and **cyl7** only.

3 In the **Settings** window for Difference, locate the **Difference** section.
4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.
5 Select the objects **cyl2**, **cyl8**, **cyl6**, and **cyl4** only.
6 Right-click **Difference 1 (dif1)** and choose **Build Selected**.
7 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.
2 Click the **Wireframe Rendering** button on the **Graphics** toolbar.
3 In the **Settings** window for Work Plane, locate the **Plane Definition** section.
4 From the **Plane type** list, choose **Face parallel**.
5 Find the **Planar face** subsection. Select the **Active** toggle button.
6 On the object **dif1**, select Boundary 3 only.
7 Click **Show Work Plane**.

Plane Geometry

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

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.
2 In the **Settings** window for Circle, locate the **Size and Shape** section.
3 In the **Radius** text field, type 2.
4 Right-click **Circle 1 (c1)** and choose **Build Selected**.

   Last, create a circular edge to be used in the **Coil** feature as a reference edge.

5 In the **Model Builder** window, click **Geometry 1**.

Work Plane 2 (wp2)

1 On the **Geometry** toolbar, click **Work Plane**.
2 In the **Settings** window for Work Plane, locate the **Plane Definition** section.
3 From the **Plane type** list, choose **Face parallel**.
4 Find the **Planar face** subsection. Select the **Active** toggle button.
5 On the object **dif1**, select Boundary 13 only.
6 Click **Show Work Plane**.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for Circle, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type 8.

5 On the **Work Plane** toolbar, click **Build All**.

Created in COMSOL Multiphysics 5.2a

Magnetic Prospecting of Iron Ore Deposits

Introduction

Magnetic prospecting is one of the methods of geological exploration applicable to certain types of iron ore deposits, in particular those made up of magnetite and hematite. Estimates of center-of-mass coordinates and the spatial extent of iron-rich layers help decrease the cost of exploration. Passive magnetic prospecting relies on accurate mapping of local geomagnetic anomalies—deviations of the natural magnetostatic fields from their predicted values based on the magnetic dipole model of the Earth's field. This application investigates magnetic anomaly estimation for both surface and aerial prospecting.

Crustal magnetic anomalies may result from either induced or remnant magnetization of iron-rich rocks. Amongst all terrestrial iron minerals, magnetite has the largest magnetic permeability (up to $\mu_r = 650$ in small-grain magnetite) and the strongest naturally occurring thermo-remnant magnetization (up to $M_r = 5000$ A/m) (Ref. 1). Another mineral with relatively strong magnetization is hematite (Ref. 1). Magnetite and hematite are the major minerals of high-grade iron ore. This application uses modest values of magnetic permeability ($\mu_r = 3.5$) and remnant magnetization ($M_r = 60$ A/m) that are typical for terrestrial magnetite with a grain size distribution of 20–200 μm (Ref. 1). The concentration of magnetite in the iron-rich ore is assumed to be 25%.

Model Definition

Magnetostatic problems without significant electrical currents can be solved in terms of a scalar magnetic potential. Variations of the magnetic permeability or remnant magnetization cause small deviations of magnetic fields from the norm (local geomagnetic anomalies), which you can model accurately using the reduced potential formulation of the Magnetic Fields, No Currents interface. The background magnetic field is assumed to be uniform within the simulation domain. Intensity and orientation of the natural magnetic field is estimated using the data from National Geophysical Data Center of the U.S. Government (Ref. 2).

This application neglects the induced and remnant magnetization of the rocks outside the iron ore body. The iron ore bed is approximated by a uniform flat ellipsoid with a maximum vertical thickness of 100 m, a maximum North-South width of 400 m, and an East-West extent of 2000 m. Center-of-mass coordinates of the ore bed (2500, 1500) meters from the lower-left corner (200 meters above sea level) are positioned underneath the Eastern Pit of the Eagle Mountain Mine, a former Kaiser Steel Co. mining operation in Riverside County, California (Ref. 3). The specific location and shape of magnetic ore in this application are not based on any actual geological prospecting but are chosen to simulate a basic size and shape of such an ore. Figure 1 shows the model geometry.

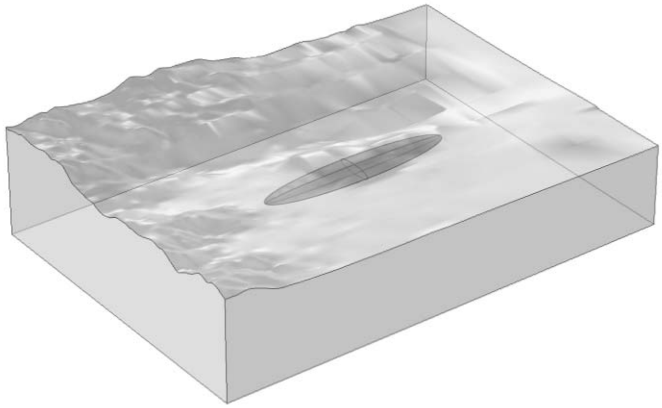


Figure 1: The model geometry.

TOPOGRAPHY

The topographical map used for this application consists of a rectangular grid containing 157-by-111 points spaced 1 arc second (1/3600°) apart in both horizontal directions. The curvature of the geoid is neglected in drawing this geometry from elevation data. The lower-left (south-west) corner of the map is located at a latitude 33.85° (North) and longitude 115.5° (West). The size of the simulation domain is 4836 m in an East-West (x-axis), 3410 m in a North-South direction (y-axis), and 1934.4 m in the up-down direction (z-axis). A satellite image of this area can be seen in (Ref. 4).

A digital elevation model (DEM) for this example can be derived from the National Elevation Dataset (NED) of the U.S. Geological Survey data center (Ref. 5). The free program MicroDEM (Ref. 6) converts the USGS elevation data into an ASCII file containing the elevation matrix, which you can import into COMSOL Multiphysics as an Interpolation function feature. From the interpolation function you can then create a geometry by using a Parametric Surface feature. Optionally, the resulting geometry can be saved in COMSOL's internal binary CAD file format (.mphbin). Importing the saved CAD file into the COMSOL Desktop is the starting point for the step-by-step instructions for this application. Figure 2 contains a plot of the resulting elevation map as it appears in

COMSOL Multiphysics.

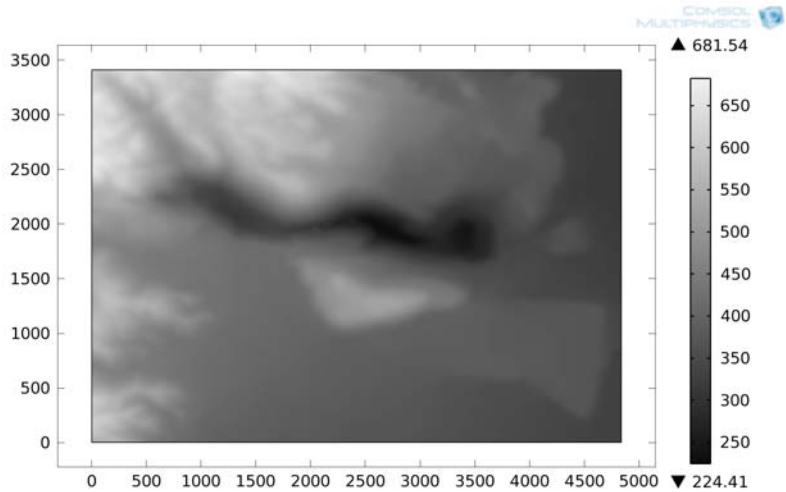


Figure 2: Elevation map of the simulation domain (boundary plot of z-coordinate on Boundary 6).

DOMAIN EQUATIONS

In a current-free region, where

$$\nabla \times \mathbf{H} = 0$$

it is possible to define a scalar magnetic potential, V_m, such that

$$\mathbf{H} = -\nabla V_m$$

This is analogous to the definition of the electric potential for static electric fields. Using the constitutive relation between the magnetic flux density and magnetic field,

$$\mathbf{B} = \mu \mathbf{H} + \mathbf{B}_r$$

together with the equation

$$\nabla \cdot \mathbf{B} = 0$$

one can derive an equation for V_m:

$$\nabla \cdot (-\mu \nabla V_m + \mathbf{B}_r) = 0$$

The reduced potential formulation used in this model splits the total magnetic potential into external and reduced parts, $V_{tot} = V_{ext} + V_{red}$, where the reduced potential V_{red} is the dependent variable:

$$\nabla \cdot (-\mu \nabla (V_{\text{ext}} + V_{\text{red}}) + \mathbf{B}_{\text{r}}) = 0$$

The external magnetic potential is more easily defined as an external magnetic field, so the equation that is used in practice is rather

$$\nabla \cdot (-\mu \nabla (V_{\text{red}}) + \mathbf{B}_{\text{r}} + \mu \mathbf{H}_{\text{ext}}) = 0$$

To simulate the background geomagnetic field, components of the external magnetic field are expressed through total intensity, magnetic declination, and inclination angles as follows:

$$\begin{aligned} \text{extHOx} &= \text{HO} \cdot \cos(\text{Incl}) \cdot \sin(\text{Decl}) \\ \text{extHOy} &= \text{HO} \cdot \cos(\text{Incl}) \cdot \cos(\text{Decl}) \\ \text{extHOz} &= -\text{HO} \cdot \sin(\text{Incl}) \end{aligned}$$

Based on the data provided by NOAA's data center (Ref. 3), inclination and declination angles for this location (N 33.85°, W 115.5°) are approximately $\text{Incl} = 59.357°$ and $\text{Decl} = 12.275°$, respectively. The magnitude of natural magnetic flux density ($B_0 = \mu_0 H_0$) is estimated as 48.163 μT.

Remnant magnetic flux density is assumed to be aligned with the local contemporary direction of geomagnetic field:

$$\begin{aligned} B_{r,x} &= B_{r,\text{ore}} \cdot \cos(\text{Incl}) \cdot \sin(\text{Decl}) \\ B_{r,y} &= B_{r,\text{ore}} \cdot \cos(\text{Incl}) \cdot \cos(\text{Decl}) \\ B_{r,z} &= -B_{r,\text{ore}} \cdot \sin(\text{Incl}) \end{aligned}$$

In general, thermo-remnant magnetization does not have to be aligned with the Earth's magnetic field.

The values of magnetic permeability and remnant flux used for the iron ore are based on the typical parameters of magnetite ore and a simple homogenization model taking into account dilution of magnetization in the ore bed:

$$\begin{aligned} \mu_{\text{r, ore}} - 1 &= (\mu_{\text{r, magn}} - 1) \cdot c_{\text{magn}} \\ \mathbf{B}_{\text{r, ore}} &= \mathbf{B}_{\text{r, magn}} \cdot c_{\text{magn}} \end{aligned}$$

Concentration of magnetite c_{magn} in the ore is assumed to be 25%.

BOUNDARY CONDITIONS

Along the exterior boundaries of the surrounding box, the perturbation of the magnetic field (H_{red}) is assumed to be tangential to the boundaries. The natural boundary condition from the equation is

$$\mathbf{n} \cdot \mathbf{B}_{red} = \mathbf{n} \cdot \mu \cdot \nabla(V_m) = 0$$

Thus, the reduced magnetic field is made tangential to the boundary by a Neumann condition on the reduced potential. Interior boundaries are modeled as continuity boundary conditions and require no user input.

The Magnetic Fields, No Currents interface automatically adds a point constraint, $V_m = 0$, using a weak term on the equation-system level if there are no boundary conditions that constrain the value of V_m (such as Magnetic potential or Zero potential). This ensures that the scalar potential V_m is uniquely defined in problems with pure Neumann boundary conditions.

Results and Discussion

Figure 3 shows deviations of the magnetic field intensity on the ground (boundary plot), and at an altitude of 1300 meters. The maximum magnetic field perturbation on the ground is 2 A/m and at 1300 meters altitude it has fallen to 0.1 A/m. In both locations the maximum is located approximately above the center of mass of the magnetite ore body. These results show that aerial magnetic prospecting may reveal the location and provide estimates of the horizontal extent of magnetite-rich deposits but also that ground-based magnetic prospecting apparently is a much more sensitive exploration tool than aerial prospecting.

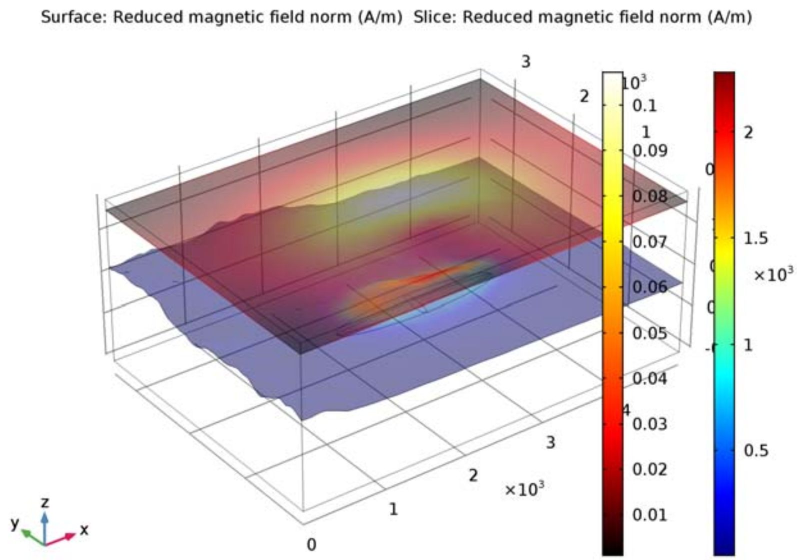


Figure 3: The slice color plot shows perturbations of magnetic intensity (norm of reduced magnetic field) at an altitude 1300 m. The boundary plot shows the same quantity on the ground.

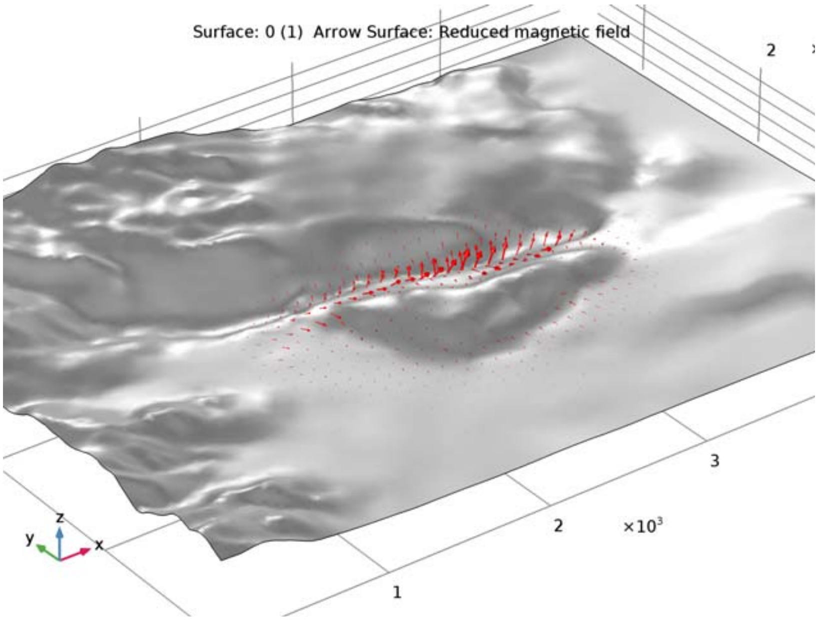


Figure 4: Arrows show the direction and relative magnitude of the reduced magnetic field on the Earth's surface.

Figure 5 shows a slice plot of the dimensionless ratio of remanent and induced magnetizations, $B_r/(\mu_0 M)$, inside the iron ore body, at an elevation of 200 m above sea level. This plot indicates that contributions of remanent and induced magnetizations to the magnetic anomaly can be comparable for realistic magnetic parameters of magnetite (Ref. 1).

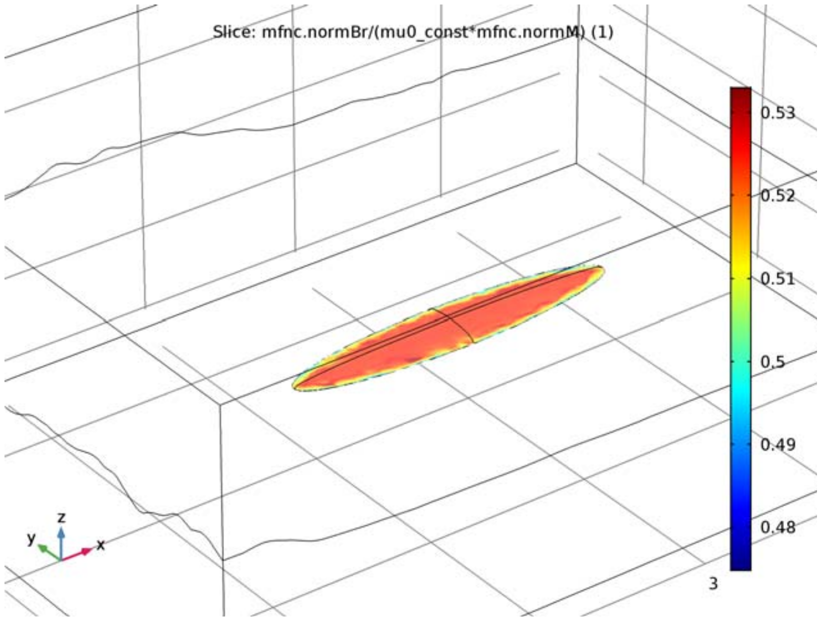


Figure 5: Slice plot of the dimensionless ratio of remanent to induced magnetizations inside the iron ore body at an elevation of 200 m above sea level. The two contributions to the magnetic anomaly are of comparable magnitude.

References

1. G. Kletetschka, P.J. Wasilewski, and P.T. Taylor, "Hematite vs. magnetite as the signature for planetary magnetic anomalies," *Physics of the Earth and Planetary Interiors*, vol. 119, pp. 259–267, 2000.

2. NOAA, National Centers for Environmental Information, http://www.ngdc.noaa.gov/geomag-web/

3. E.R. Force, "Eagle Mountain Mine – Geology of the former Kaiser Steel operation in Riverside county, California," *U.S. Geological Survey Open-File Report* 01-237.

4. Google Maps, http://maps.google.com/maps?f=q&hl=en&geocode=&q=33.865,-115.475&ie=UTF8&t=h&z=16&iwloc=addr

5. Data available from U.S. Geological Survey, National Elevation Data set, using The National Map Viewer, http://nationalmap.gov/viewer.html

6. MICRODEM Home Page, http://www.usna.edu/Users/oceano/pguth/website/microdem/microdem.htm

Application Library path: `ACDC_Module/Magnetostatics/magnetic_prospecting`

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>Magnetic Fields, No Currents (mfnc)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
mur_magn	3.5	3.5	Relative permittivity of magnetite
c_magn	0.25	0.25	Magnetite concentration in ore
Mr_magn	60[A/m]	60 A/m	Remanent magnetization of magnetite

Name	Expression	Value	Description
H0	48163[nT]/ mu0_const	38.327 A/m	Geoelectric field
Incl	59.357[deg]	1.036 rad	Local inclination
Decl	12.275[deg]	0.21424 rad	Local declination

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Global Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Br_magn	Mr_magn*mu0_const	T	Remanent flux density of magnetite
Br	Br_magn*c_magn	T	Remanent flux density of ore
Gx	cos(Incl)*sin(Decl)		Geomagnetic field direction, x-component
Gy	cos(Incl)*cos(Decl)		Geomagnetic field direction, y-component
Gz	-sin(Incl)		Geomagnetic field direction, z-component
mur_ore	1+(mur_magn-1)*c_magn		Model for the relative permittivity of ore

GEOMETRY 1

The geometry obtained from the heightmap is available in an MPHBIN-file (COMSOL Multiphysics' native CAD file format) that can be imported in the geometry node.

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the application's Application Libraries folder and double-click the file `magnetic_prospecting.mphbin`.

5 Click **Import**.

Create an ellipsoid that models the iron ore deposit.

Ellipsoid 1 (elp1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Ellipsoid**.
2 In the **Settings** window for Ellipsoid, locate the **Size and Shape** section.
3 In the **a-semiaxis** text field, type `1000`.
4 In the **b-semiaxis** text field, type `200`.
5 In the **c-semiaxis** text field, type `50`.
6 Locate the **Position** section. In the **x** text field, type `2500`.
7 In the **y** text field, type `1500`.
8 In the **z** text field, type `200`.
9 Click **Build All Objects**.

   Activate the transparency to verify that the deposit is in the correct position.

10 Click the **Transparency** button on the **Graphics** toolbar.

MATERIALS

Use one material for the background (air and nonmagnetic rocks) and another for the iron ore. For the ore, you specify the relative permeability as a material property but add the remanent magnetization in the physics settings.

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for Material, locate the **Material Contents** section.
3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permeability	mur	1	1	Basic

4 Right-click **Component 1 (comp1)>Materials>Material 1 (mat1)** and choose **Rename**.
5 In the **Rename Material** dialog box, type `Background Material` in the **New label** text field.
6 Click **OK**.

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2 Select Domain 3 only.

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

4 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Relative permeability	mur	mur_ore	1	Basic

5 Right-click **Component 1 (comp1)>Materials>Material 2 (mat2)** and choose **Rename**.

6 In the **Rename Material** dialog box, type Ore in the **New label** text field.

7 Click **OK**.

MAGNETIC FIELDS, NO CURRENTS (MFNC)

Magnetic Flux Conservation 2

1 On the **Physics** toolbar, click **Domains** and choose **Magnetic Flux Conservation**.

2 Select Domain 3 only.

3 In the **Settings** window for Magnetic Flux Conservation, locate the **Magnetic Field** section.

4 From the **Constitutive relation** list, choose **Remanent flux density**.

5 Specify the $\mathbf{B}_r$ vector as

Br*Gx	x
Br*Gy	y
Br*Gz	z

Specify the local geomagnetic field as the background field for the problem.

6 In the **Model Builder** window, click **Magnetic Fields, No Currents (mfnc)**.

7 In the **Settings** window for Magnetic Fields, No Currents, locate the **Background Magnetic Field** section.

8 From the **Solve for** list, choose **Reduced field**.

9 Specify the $\mathbf{H}_b$ vector as

H0*Gx	x
H0*Gy	y
H0*Gz	z

External Magnetic Flux Density 1

1 On the **Physics** toolbar, click **Boundaries** and choose **External Magnetic Flux Density**.

   Apply **External Magnetic Flux Density** as the boundary condition instead of **Magnetic Insulation**. The former operates only on the relative field, while the latter sets a condition for the total field.

2 Select Boundaries 1–5, 7–9, 18, and 19 only.

STUDY 1

Disable the automatic generation of the default plots. You will manually create the plots after the solution.

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.

4 On the **Home** toolbar, click **Compute**.

RESULTS

In the **Model Builder** window, expand the **Results** node.

Data Sets

Create a second solution with a selection active only on the boundary corresponding to the Earth's surface. You will use this solution when plotting surface data.

Study 1/Solution 1 (2) (sol1)

On the **Results** toolbar, click **More Data Sets** and choose **Solution**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 Select Boundary 6 only.

3D Plot Group 1

On the **Results** toolbar, click **3D Plot Group**.

Surface 1

1 In the **Model Builder** window, right-click **3D Plot Group 1** and choose **Surface**.

2 In the **Settings** window for Surface, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Solution 1 (2) (sol1)**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields, No Currents>Magnetic>mfnc.normredH - Reduced magnetic field norm**.

3D Plot Group 1

The plot of the relative magnetic field at the altitute of 1300 m can be obtained with a simple slice plot.

Slice 1

1 In the **Model Builder** window, under **Results** right-click **3D Plot Group 1** and choose **Slice**.

2 In the **Settings** window for Slice, locate the **Plane Data** section.

3 From the **Plane** list, choose **xy-planes**.

4 From the **Entry method** list, choose **Coordinates**.

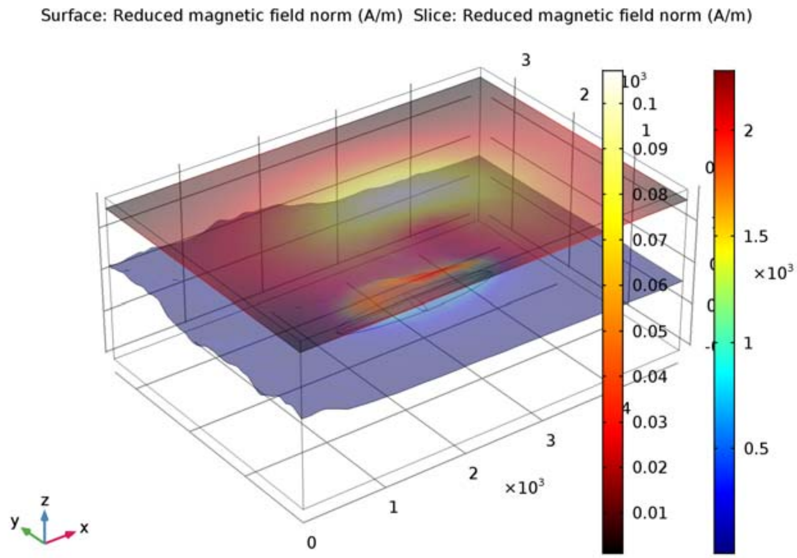
5 In the **z-coordinates** text field, type 1300.

6 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields, No Currents>Magnetic>mfnc.normredH - Reduced magnetic field norm**.

7 Locate the **Coloring and Style** section. From the **Color table** list, choose **Thermal**.

8 On the **3D Plot Group 1** toolbar, click **Plot**.

The plot shows the alteration of the geomagnetic field due to the presence of a magnetized material. Note how the relative magnetic field intensity at ground level is about 20 times larger than at 1300 m.



9 Click the **Transparency** button on the **Graphics** toolbar to return to the default state.

Next, add another plot group to visualize the vector field.

3D Plot Group 2

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

Add a uniform gray surface representing the ground.

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

3 In the **Settings** window for 3D Plot Group, locate the **Data** section.

4 From the **Data set** list, choose **Study 1/Solution 1 (2) (sol1)**.

Surface 1

1 In the **Model Builder** window, under **Results>3D Plot Group 2** click **Surface 1**.

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

3 In the **Expression** text field, type `0`.

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

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

3D Plot Group 2

Now plot the relative magnetic field at ground level.

Arrow Surface 1

1 In the **Model Builder** window, under **Results** right-click **3D Plot Group 2** and choose **Arrow Surface**.

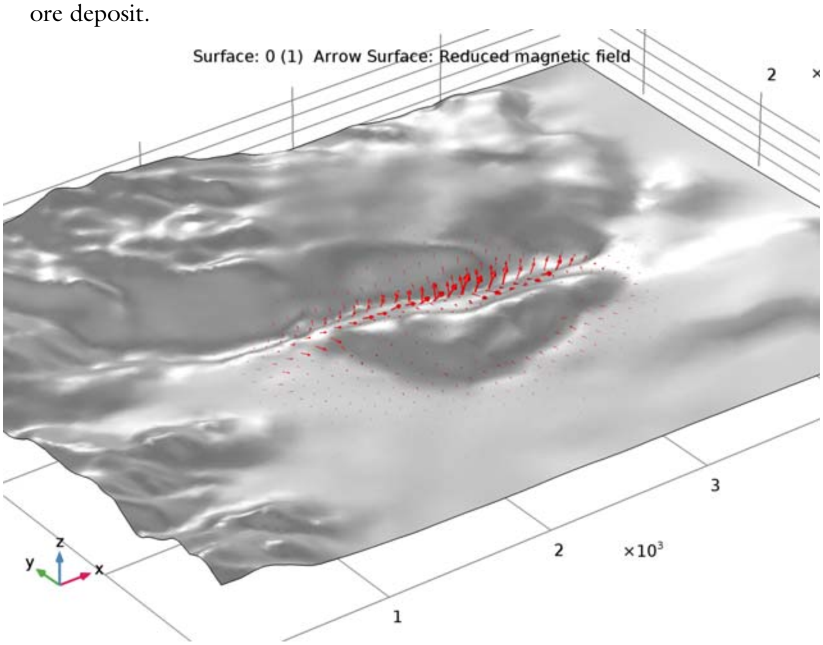
2 In the **Settings** window for Arrow Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields, No Currents>Magnetic>mfnc.redHx,...,mfnc.redHz - Reduced magnetic field**.

3 Locate the **Coloring and Style** section. In the **Number of arrows** text field, type 2000.

4 On the **3D Plot Group 2** toolbar, click **Plot**.

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

The plot shows the vector field for the magnetic field perturbation caused by the iron ore deposit.



Finally, plot the ratio between the remanent flux density and the induced magnetization.

3D Plot Group 3

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

Slice 1

1 In the **Model Builder** window, right-click **3D Plot Group 3** and choose **Slice**.

2 In the **Settings** window for Slice, locate the **Plane Data** section.

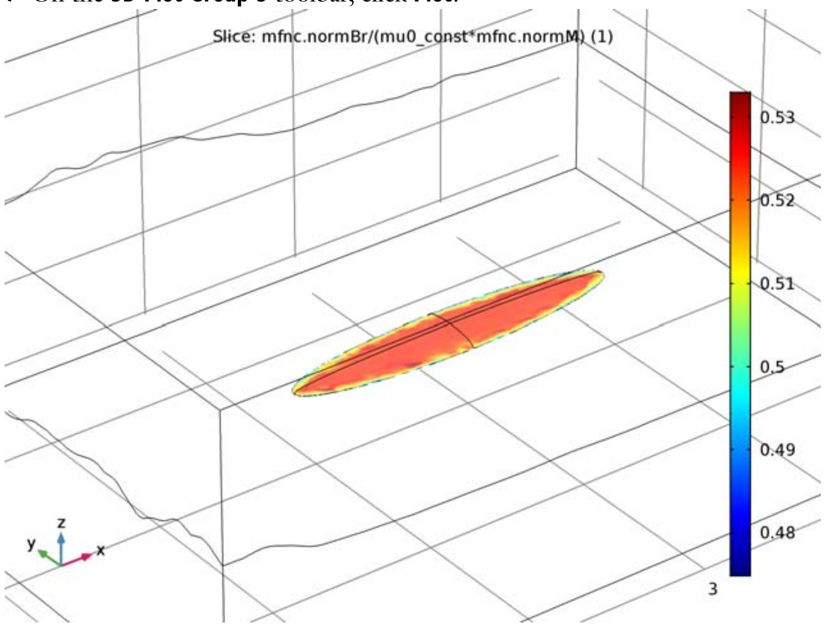
3 From the **Plane** list, choose **xy-planes**.

4 From the **Entry method** list, choose **Coordinates**.

5 In the **z-coordinates** text field, type `200`.

6 Locate the **Expression** section. In the **Expression** text field, type `mfnc.normBr/(mu0_const*mfnc.normM)`.

7 On the **3D Plot Group 3** toolbar, click **Plot**.



The plot shows that the two contributions are comparable in magnitude.

8 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Multi-Turn Coil Above an Asymmetric Conductor Plate

Introduction

This model solves the Testing Electromagnetic Analysis Methods (TEAM) problem 7, "Asymmetrical Conductor with a Hole"—a benchmark problem concerning the calculation of eddy currents and magnetic fields produced when an aluminum conductor is placed asymmetrically above a multi-turn coil carrying a sinusoidally varying current. The simulation results at specified positions in space are compared with measured data from the literature and agreement is shown.

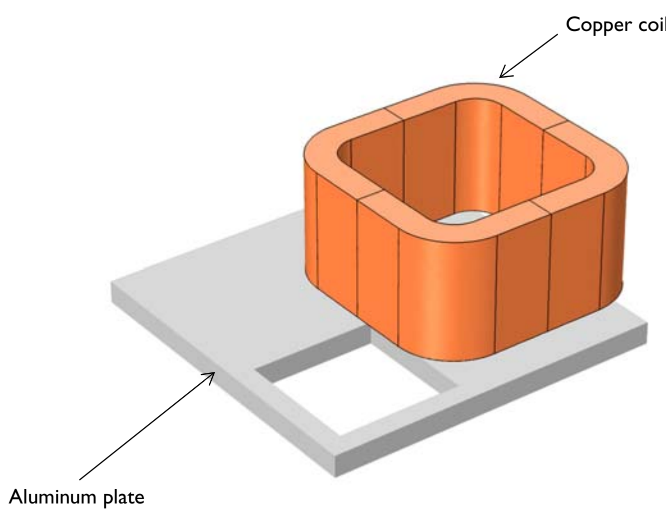


Figure 1: A multi-turn coil placed asymmetrically over an aluminum plate with a hole.

Model Definition

Because the geometry has no symmetries, the problem must be solved for the entire geometry. As shown in Figure 1, the geometry consists of a coil placed asymmetrically above a thick aluminum conductor with an eccentric square hole. The coil has 2742 turns and carries a sinusoidally varying current of 1 A/turn. The problem is to compute the magnetic field and the eddy currents induced in the conductor for coil currents of frequencies 50 Hz and 200 Hz, and to compare the simulation results along specified locations in space with experimental data given in Ref. 1.

Results and Discussion

Figure 2 visualizes the induced current at 50 Hz using a combined surface and arrow plot. The black arrows show the current density in the conductor while the red arrows indicate the coil current direction is shown. Figure 3 shows the corresponding results at 200 Hz.

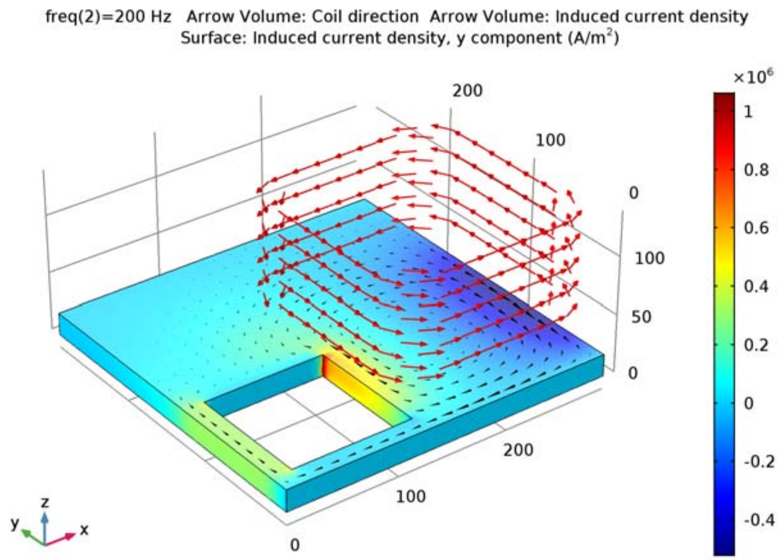


Figure 2: A 3D surface plot of the y-component of the induced current density, J_y (A/m^2) in the conductor combined with the arrow volume plots of the coil current direction and the induced current density in the conductor. This simulation results are at 50 Hz.

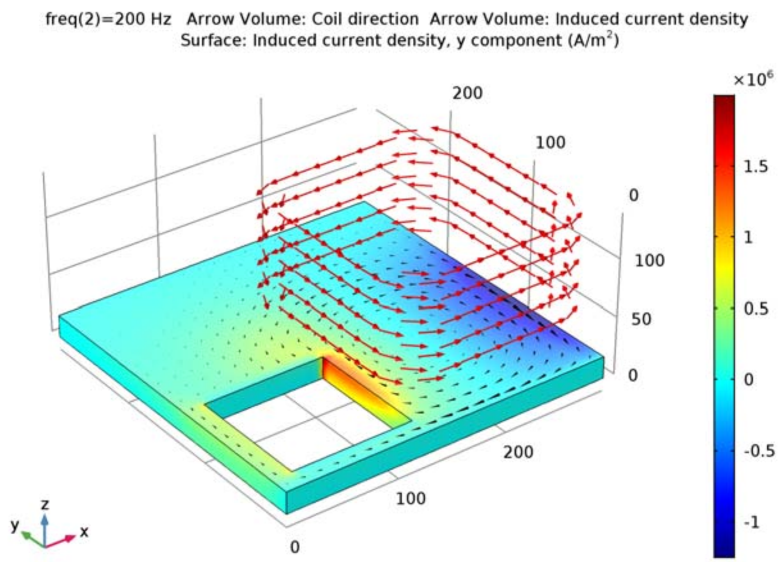


Figure 3: A 3D surface plot of the y-component of the induced current density in the conductor combined with the arrow volume plots of the coil current direction and the induced current density in the conductor. This simulation results are at 200 Hz.

Figure 4 and Figure 5 show comparisons between the simulation and the measured data. In Figure 4, the z-component values of the magnetic flux density, B_z (SI unit: mT), at 50 Hz and 200 Hz are compared with measured data from Ref. 1. These results are compared for $0 \leq x \leq 288$ mm at $y = 72$ mm and $z = 34$ mm. In Figure 5, similarly, the y-component values of the induced current density, J_y (SI unit: A/m^2) at 50 Hz and 200 Hz are compared with the corresponding measured data. These results are compared for $0 \leq x \leq 288$ mm at $y = 72$ mm and $z = 19$ mm. As is evident from the plots, there is close agreement between the simulation results and the measured data.

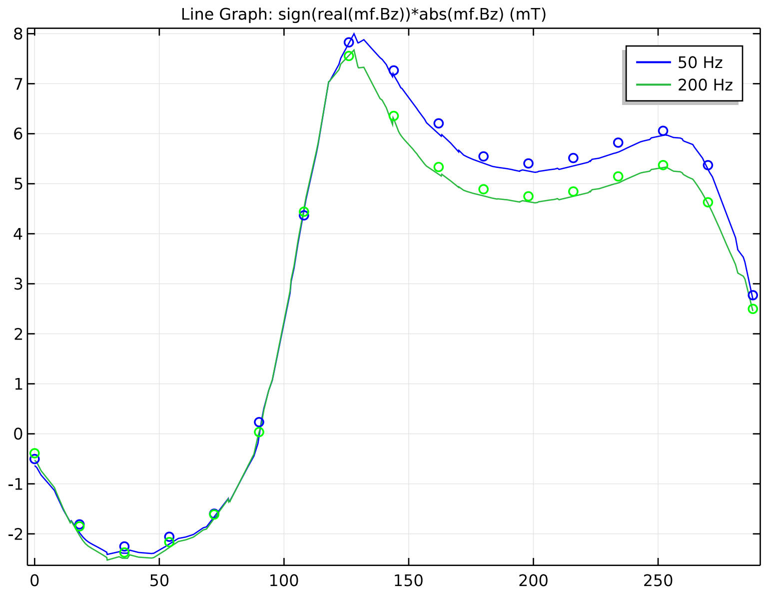


Figure 4: z-components of the flux densities, B_z (mT) for $0 \leq x \leq 288$ mm at $y = 72$ mm, $z = 34$ mm for 50 Hz and 200 Hz. The solid lines are simulations results while the circles are the corresponding measured data from Ref. 1.

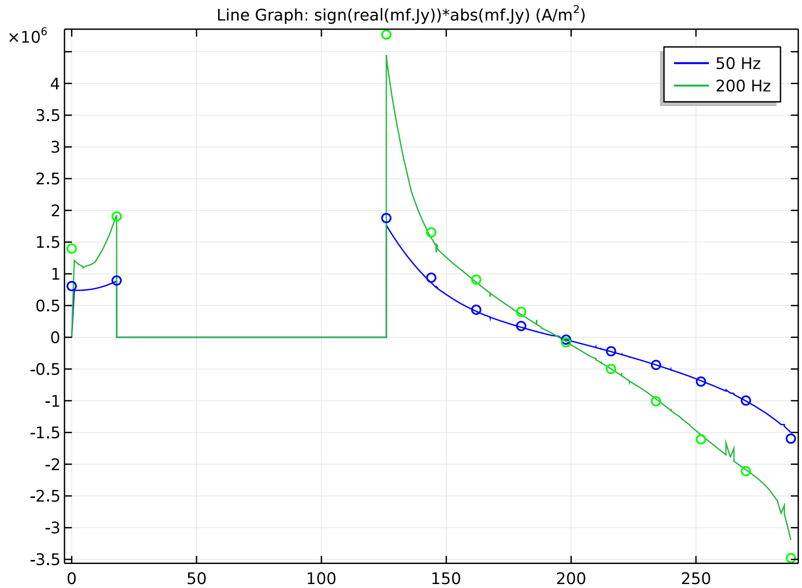


Figure 5: y-components of the induced current densities, J_y (A/m^2) for $0 \leq x \leq 288$ mm at $y = 72$ mm, $z = 19$ mm for 50 Hz and 200 Hz. The solid lines are simulation results while the circles are the corresponding measured data from Ref. 1.

Notes About the COMSOL Implementation

To compute the coil currents, model the coil using a Coil feature with a Geometry Analysis subfeature combined with a Coil Geometry Analysis study step. Compute the eddy currents and the nonuniform magnetic field using a Frequency Domain study step for the frequencies 50 Hz and 200 Hz.

Reference

1. K. Fujiwara and T. Nakata, "Results for Benchmark Problem 7 (Asymmetrical Conductor with a Hole)," *Int. J. Comput. and Math. in Electr. and Electron. Eng.*, vol. 9, no. 3, pp. 137–154, 1990.

Application Library path: `ACDC_Module/Inductive_Devices_and_Coils/multiturn_coil_asymmetric_conductor`

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>Magnetic Fields (mf)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Empty Study**.
6. Click **Done**.

GEOMETRY 1

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

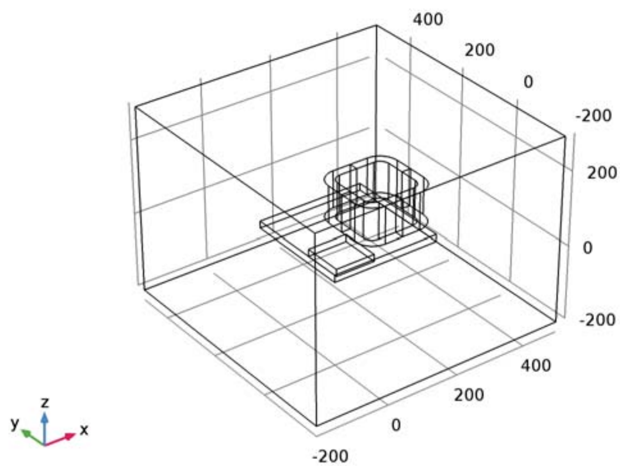
1. On the **Geometry** toolbar, click **Insert Sequence**.
2. Browse to the application's Application Libraries folder and double-click the file `multiturn_coil_asymmetric_conductor_geom_sequence.mph`.

Form Union (fin)

1. On the **Geometry** toolbar, click **Build All**.
2. Click the **Zoom Extents** button on the **Graphics** toolbar.

   The model geometry is now complete. Choose wireframe rendering to get a better view of the interior parts.

3 Click the **Wireframe Rendering** button on the **Graphics** toolbar.



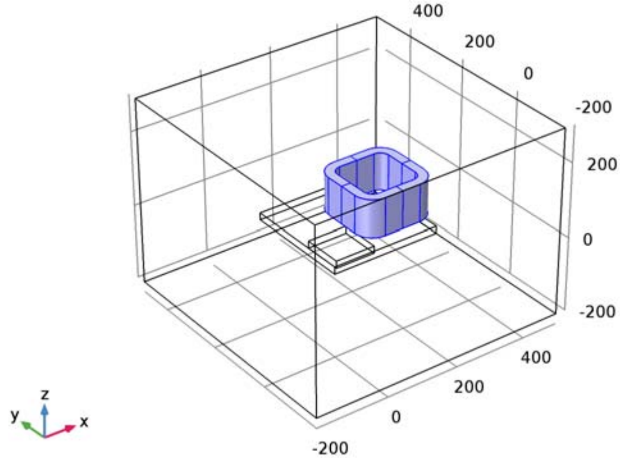
DEFINITIONS

Define domain and boundary selections for the coil and the conductor.

Explicit 1

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

2 Select Domains 3–6 only.



3 Right-click **Explicit 1** and choose **Rename**.

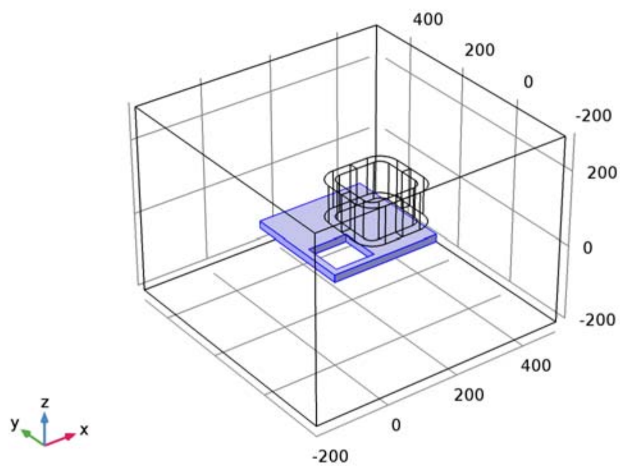
4 In the **Rename Explicit** dialog box, type `Coil` in the **New label** text field.

5 Click **OK**.

Explicit 2

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

2 Select Domain 2 only.



3 Right-click **Explicit 2** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Conductor` in the **New label** text field.

5 Click **OK**.

View 1

Hide the outer geometry boundaries to visualize the results only in the coil and the conductor domains.

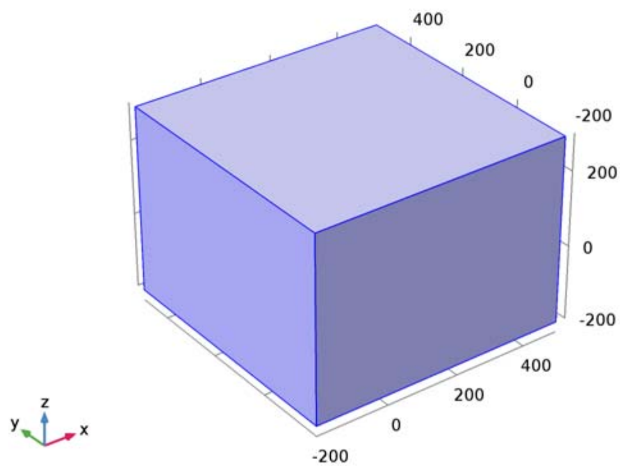
Hide for Physics 1

1 On the **View 1** toolbar, click **Hide for Physics**.

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

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

4 Select Boundaries 1–5 and 52 only.



MAGNETIC FIELDS (MF)

Now set up the physics. **Ampère's Law** automatically applies on all the domains. Add the **Coil** feature to model the coil. This will automatically overwrite the **Ampère's Law** feature.

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Magnetic Fields (mf)** and choose **Group by Space Dimension**.

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

Coil 1

1 On the **Physics** toolbar, click **Domains** and choose **Coil**.

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

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

4 Locate the **Coil** section. From the **Conductor model** list, choose **Homogenized multi-turn**.

5 From the **Coil type** list, choose **Numeric**.

6 Locate the **Homogenized Multi-Turn Conductor** section. In the N text field, type 2742.

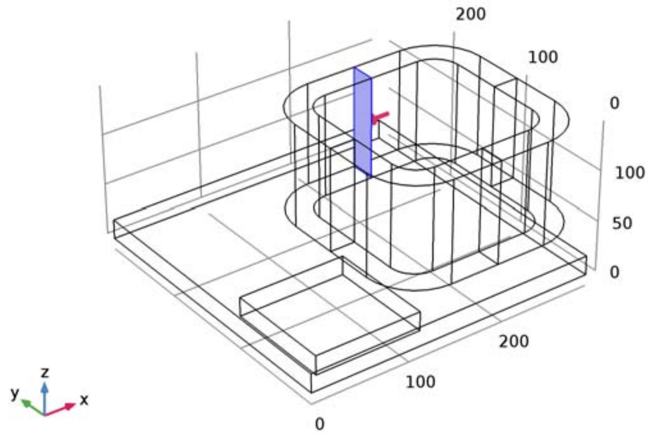
Apply input on a cross-sectional coil boundary.

7 In the **Model Builder** window, expand the **Coil 1** node.

Input 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Magnetic Fields (mf)>Domains>Coil 1>Geometry Analysis 1** node, then click **Input 1**.

2 Select Boundary 37 only.



MATERIALS

Apply the material properties of air in all domains.

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

MATERIALS

Air (mat1)

The conductivity of air in the Material Library is zero. Change the conductivity to 10 S/m to improve the stability of the solution. The error caused by this small conductivity is negligible.

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.

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

3 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Electrical conductivity	sigma	`10[S/m]`	S/m	Basic

Assign the material properties of the conductor. This will overwrite the material properties of air.

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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

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

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

Property	**Name**	**Value**	**Unit**	**Property group**
Relative permeability	mur	`1`	1	Basic
Electrical conductivity	sigma	`3.526e7`	S/m	Basic
Relative permittivity	epsilonr	`1`	1	Basic

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

6 Right-click **Component 1 (comp1)>Materials>Material 2 (mat2)** and choose **Rename**.

7 In the **Rename Material** dialog box, type `Aluminum` in the **New label** text field.

8 Click **OK**.

MESH 1

To get the accurate results, mesh the aluminum plate finer than the rest of the geometry.

Size 1

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

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 text field, type `12`.

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 **Coil**.

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

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

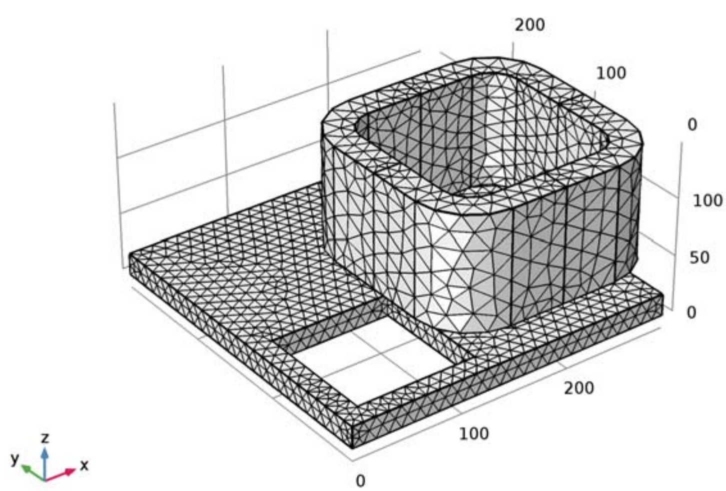
7 In the associated text field, type `20`.

Free Tetrahedral 1

1 Right-click **Mesh 1** and choose **Free Tetrahedral**.

2 In the **Settings** window for Free Tetrahedral, click **Build Selected**.

The mesh should look like that shown in the figure below.



Solve the problem in the frequency domain for 50 Hz and 200 Hz. Note that you need to add a **Coil Geometry Analysis** study step before the Frequency Domain step.

STUDY 1

Coil Geometry Analysis

On the **Study** toolbar, click **Study Steps** and choose **Other>Coil Geometry Analysis**.

Frequency Domain

On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>Frequency Domain**.

Step 2: Frequency Domain

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

2 In the **Frequencies** text field, type `50 200`.

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

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

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

6 On the **Study** toolbar, click **Compute**.

RESULTS

In the **Model Builder** window, expand the **Results** node.

Study 1/Solution 1 (sol1)

Create separate data sets for the conductor and coil domains. This is useful for visualizing the results in different domains independently.

1 In the **Model Builder** window, expand the **Results>Data Sets** node.

2 Right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 From the **Selection** list, choose **Conductor**.

5 Select the **Propagate to lower dimensions** check box.

6 Right-click **Selection** and choose **Rename**.

7 In the **Rename Selection** dialog box, type `Conductor` in the **New label** text field.

8 Click **OK**.

Study 1/Solution 1 (1) (sol1)

In the **Model Builder** window, under **Results>Data Sets** right-click **Study 1/Solution 1 (1) (sol1)** and choose **Duplicate**.

Selection

1. On the **Results** toolbar, click **Selection**.
2. In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. From the **Selection** list, choose **Coil**.
5. Select the **Propagate to lower dimensions** check box.
6. Right-click **Selection** and choose **Rename**.
7. In the **Rename Selection** dialog box, type `Coil` in the **New label** text field.
8. Click **OK**.

   To reproduce the plot shown in Figure 2, create a 3D plot group and add an arrow volume plot of the coil current direction in the coil domain. Add an arrow volume plot of the induced current on the conductor surface and a surface plot of the y component of the induced current density to the same plot group.

3D Plot Group 1

1. On the **Results** toolbar, click **3D Plot Group**.
2. In the **Settings** window for 3D Plot Group, locate the **Data** section.
3. From the **Data set** list, choose **Study 1/Solution 1 (3) (sol1)**.

Arrow Volume 1

1. Right-click **3D Plot Group 1** and choose **Arrow Volume**.
2. In the **Settings** window for Arrow Volume, locate the **Data** section.
3. From the **Data set** list, choose **Study 1/Solution 1 (4) (sol1)**.
4. From the **Parameter value (freq (Hz))** list, choose **50**.
5. Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields>Coil parameters>mf.coil1.eCoilx,..., mf.coil1.eCoilz - Coil direction**.
6. Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `10`.
7. Find the **y grid points** subsection. In the **Points** text field, type `10`.
8. Find the **z grid points** subsection. In the **Points** text field, type `5`.
9. Locate the **Coloring and Style** section. Select the **Scale factor** check box.
10. In the associated text field, type `20`.

3D Plot Group 1

In the **Model Builder** window, under **Results** right-click **3D Plot Group 1** and choose **Arrow Volume**.

Arrow Volume 2

1 In the **Settings** window for Arrow Volume, locate the **Data** section.

2 From the **Data set** list, choose **Study 1/Solution 1 (3) (sol1)**.

3 From the **Parameter value (freq (Hz))** list, choose **50**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields>Currents and charge>mf.Jix,...,mf.Jiz - Induced current density**.

5 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type 20.

6 Find the **y grid points** subsection. In the **Points** text field, type 20.

7 Find the **z grid points** subsection. From the **Entry method** list, choose **Coordinates**.

8 In the **Coordinates** text field, type 19.

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

10 From the **Arrow type** list, choose **Cone**.

3D Plot Group 1

Right-click **3D Plot Group 1** and choose **Surface**.

Surface 1

1 In the **Settings** window for Surface, locate the **Data** section.

2 From the **Data set** list, choose **Study 1/Solution 1 (3) (sol1)**.

3 From the **Parameter value (freq (Hz))** list, choose **50**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields>Currents and charge>Induced current density>mf.Jiy - Induced current density, y component**.

5 On the **3D Plot Group 1** toolbar, click **Plot**.

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

3D Plot Group 1

Duplicate the plot group you just created and modify the new plot group to generate the plot shown in Figure 3. This figure is similar to the Figure 2 except the frequency is 200 Hz.

1 Right-click **3D Plot Group 1** and choose **Duplicate**.

Arrow Volume 1

1 In the **Model Builder** window, expand the **3D Plot Group 2** node, then click **Arrow Volume 1**.

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

3 From the **Parameter value (freq (Hz))** list, choose **200**.

Arrow Volume 2

1 In the **Model Builder** window, under **Results>3D Plot Group 2** click **Arrow Volume 2**.

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

3 From the **Parameter value (freq (Hz))** list, choose **200**.

Surface 1

1 In the **Model Builder** window, under **Results>3D Plot Group 2** click **Surface 1**.

2 In the **Settings** window for Surface, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **200**.

4 On the **3D Plot Group 2** toolbar, click **Plot**.

To generate the plots shown in Figure 4 and Figure 5, begin by creating two cut-line data sets.

Cut Line 3D 1

1 On the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for Cut Line 3D, locate the **Line Data** section.

3 In row **Point 1**, set **y** to 72 and **z** to 34.

4 In row **Point 2**, set **x** to 288, **y** to 72, and **z** to 34.

Cut Line 3D 2

1 Right-click **Cut Line 3D 1** and choose **Duplicate**.

Select the value of z slightly below the surface of the conductor.

2 In the **Settings** window for Cut Line 3D, locate the **Line Data** section.

3 In row **Point 1**, set **z** to 18.99.

4 In row **Point 2**, set **z** to 18.99.

Import the experimental data of Bz and Jy to tables.

Table 1

1 On the **Results** toolbar, click **Table**.

2 In the **Settings** window for Table, locate the **Data** section.

3 Click **Import**.

4 Browse to the application's Application Libraries folder and double-click the file `multiturn_coil_asymmetric_conductor_table1.txt`.

Table 2

1 On the **Results** toolbar, click **Table**.

2 In the **Settings** window for Table, locate the **Data** section.

3 Click **Import**.

4 Browse to the application's Application Libraries folder and double-click the file `multiturn_coil_asymmetric_conductor_table2.txt`.

Plot the z-component of the magnetic flux density, `Bz`, along the lines defined by the cut-line data sets you just created.

1D Plot Group 3

On the **Results** toolbar, click **1D Plot Group**.

Line Graph 1

1 On the **1D Plot Group 3** toolbar, click **Line Graph**.

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

3 From the **Data set** list, choose **Cut Line 3D 1**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `sign(real(mf.Bz))*abs(mf.Bz)`.

5 From the **Unit** list, choose **mT**.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
50 Hz
200 Hz

1D Plot Group 3

Add a plot for the experimental value of `Bz` at 50 Hz to the same plot group using a table graph.

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

Table Graph 1

1 On the **1D Plot Group 3** toolbar, click **Table Graph**.

2 In the **Settings** window for Table Graph, locate the **Data** section.

3 From the **x-axis data** list, choose **Column 1**.

4 From the **Plot columns** list, choose **Manual**.

5 In the **Columns** list, select **x [mm] B**.

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

7 From the **Color** list, choose **Blue**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

9 From the **Positioning** list, choose **In data points**.

Duplicate the table graph you created above and modify to plot the experimental value of `Bz` at 200 Hz.

Table Graph 2

1 Right-click **Table Graph 1** and choose **Duplicate**.

2 In the **Settings** window for Table Graph, locate the **Data** section.

3 In the **Columns** list, select **z(x,72,34) at 50Hz Bz(x,72,34)at 200Hz**.

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

5 On the **1D Plot Group 3** toolbar, click **Plot**.

Compare the resulting plot with that in Figure 4.

1D Plot Group 3

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 3** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog box, type `Bz(x,72,34)` in the **New label** text field.

3 Click **OK**.

Plot the *y*-component of the current density on the surface of the conductor, `Jy`, along the line you specified earlier. Use an approach similar to the one you used to generate Figure 4.

1D Plot Group 4

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

Line Graph 1

1 On the **1D Plot Group 4** toolbar, click **Line Graph**.

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

3 From the **Data set** list, choose **Cut Line 3D 2**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `sign(real(mf.Jy))*abs(mf.Jy)`.

5 Locate the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
50 Hz
200 Hz

1D Plot Group 4

In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

Table Graph 1

1 On the **1D Plot Group 4** toolbar, click **Table Graph**.

2 In the **Settings** window for Table Graph, locate the **Data** section.

3 From the **Table** list, choose **Table 2**.

4 From the **x-axis data** list, choose **x**.

5 From the **Plot columns** list, choose **Manual**.

6 In the **Columns** list, select **[mm] Jy(x**.

7 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

8 From the **Color** list, choose **Blue**.

9 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

10 From the **Positioning** list, choose **In data points**.

Table Graph 2

1 Right-click **Table Graph 1** and choose **Duplicate**.

2 In the **Settings** window for Table Graph, locate the **Data** section.

3 In the **Columns** list, select **,72,19) at 50Hz Jy(x,72,19)at 200Hz**.

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

5 On the **1D Plot Group 4** toolbar, click **Plot**.

The plot should look like that shown in Figure 5.

1D Plot Group 4

1. In the **Model Builder** window, under **Results** right-click **1D Plot Group 4** and choose **Rename**.
2. In the **Rename 1D Plot Group** dialog box, type `Jy(x,72,19)` in the **New label** text field.
3. Click **OK**.

Created in COMSOL Multiphysics 5.2a

Mutual Inductance and Induced Currents Between Single-Turn Coils

Introduction

The mutual inductance between a primary and secondary coil consisting of a single turn in a concentric coplanar arrangement is computed using a DC (steady-state) model and compared against the analytic solution. The induced currents in the secondary coil are computed using an AC (frequency-domain) analysis. The relationship between the AC induced currents and the DC inductance is discussed.

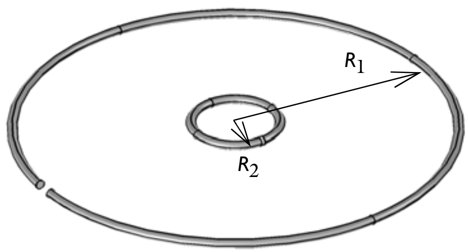


Figure 1: Two concentric coplanar single turn loops of wire. The outer is the primary coil, and has a prescribed current flowing through it.

Model Definition

The physical situation being modeled is shown in Figure 1. A prescribed current of 1 A is flowing through a single-turn coil of radius R_1 = 100 mm. A secondary coil, R_2 = 10 mm is concentric with the primary, and in the same plane. The wire radius is r_0 = 1mm. The coils are here modeled in the 2D axis-symmetric space, assuming no physical variation around the center line.

In the limit as $R_1 >> R_2 >> r_0$, the analytic expression for the mutual inductance between the two coils is:

$$M = \frac{\pi\mu_0 {R_2}^2}{2R_1}$$

where μ_0 is the permeability of free space. This analytic expression is used to verify the accuracy of the DC model.

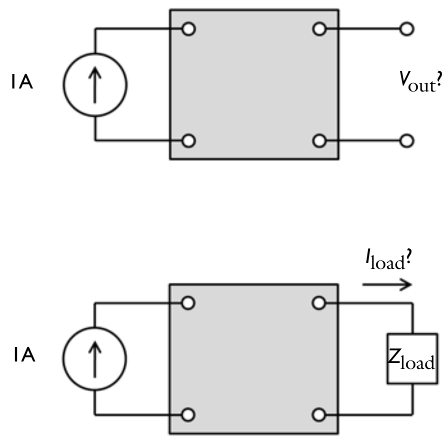


Figure 2: The concentric coils can be considered as a four-terminal device. The output can either be an open circuit, or a load can be applied.

Another way to consider this system is as a four-terminal device, as shown in Figure 2. A known current applied at the input terminals of the device, the primary, induces a voltage difference across the output terminals, the secondary. The objective of the AC model is to compute the voltage difference at the output for the open circuit case, and the induced currents for the closed circuit case.

The two concentric coils are modeled in a 2D axisymmetric sense, as shown schematically in Figure 3. The modeling domain is surrounded by a region of infinite elements, which is a way to truncate a domain which stretches to infinity. Although the thickness of the Infinite Element Domain is finite, it can be thought of as a domain of infinite extent.

The coils are both modeled using the **Coil** feature, which can be thought of as introducing an excitation across an infinitesimal slit in an otherwise continuous torus. Since each coil has a single turn and is made up of conductive material, the **Single conductor** model is used in the Coil feature. The feature can be used to excite the coil in all cases: the open circuit case, the closed torus case, as well as to model an external load. The primary coil is excited by specifying a current of 1 A.

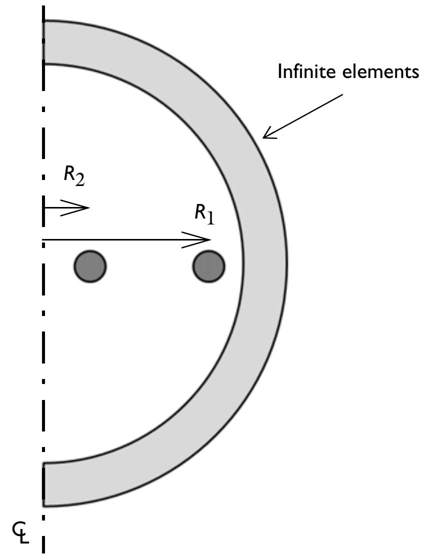


Figure 3: A schematic representation of the 2D axisymmetric model of the concentric coils.

Although induced currents exist only if there is some variation in the driving current with respect to time, it is still possible to evaluate the mutual inductance for this case from a DC analysis. The mutual inductance is defined as the total magnetic flux **B**, passing through a surface whose edges are defined by the secondary coil. That is:

$$L_{12} = \frac{\iint_{S_2} \mathbf{B} \cdot \mathbf{n} dS}{I_1}$$

Where I_1 is the current passing through the primary coil, **n** is the vector normal to the surface, and the integral is taken over the surface defined by the coil. Since the **B**-field is computed from the magnetic vector potential:

$$\mathbf{B} = \nabla \times \mathbf{A}$$

It is possible to use Stokes' theorem, which states that a surface integral of the curl of a field equals the line integral over the rim of the surface:

$$L_{12} = \frac{\iint\limits_{S_2} (\nabla \times \mathbf{A}) \cdot \mathbf{n}\, dS}{I_1} = \frac{\oint\limits_{\Gamma_S} \mathbf{A} \cdot \mathbf{t}\, dl}{I_1}$$

Where **t** is the unit tangent vector around the rim of the surface.

For a 2D axisymmetric model, these equations reduce to a line integral of the z-component of the **B**-field, or a point integral of the magnetic scalar potential at the center point of the secondary coil.

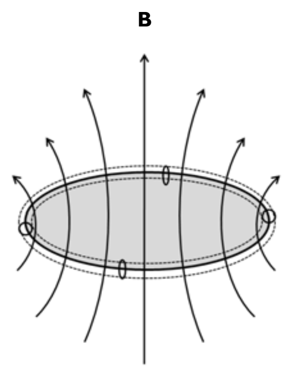


Figure 4: The mutual inductance in the secondary coil can be evaluated by taking the surface integral of the magnetic flux through the coil, or the path integral of the magnetic vector potential.

For the AC case, a 1 kHz sinusoidally time-varying current is driving the primary coil. This can either induce currents in the secondary coil or induce a voltage difference if the coil is being modeled as an open circuit.

The secondary coil uses the Coil feature to model both the open circuit and the closed circuit case. To model the open circuit case, the current through the coil is specified to be 0 A. The Coil feature introduces a coil voltage that causes no current to flow.

On the other hand, to model the closed circuit case, the voltage drop across the coil is fixed at 0 V. Although this seems to imply a short circuit, the reactance of the copper coil is inherently included, so the case being modeled is analogous to a closed continuous loop of wire.

The mutual inductance can also be evaluated in the frequency domain. The line integral of the magnetic flux can again be evaluated. However, it is no longer possible to use a point evaluation at the center point of the coil since the induced currents in the secondary also induce fields. The effects of these fields must be averaged by taking a surface integral of the magnetic vector potential over the cross section of the wire and dividing this by the area of the wire.

In the AC case, there is no analytic solution to compare against. At any non-zero frequency, capacitive effects start to appear, and the skin effect also starts to alter the effective resistance of the coils. The magnitude of these effects can only be evaluated with a frequency domain model. Although the DC case does provide good predictions of the behavior at low frequencies, it cannot completely predict behavior at higher frequencies. As additional physical objects such as cores are introduced, the need for a frequency domain model for accurate prediction becomes greater.

Results and Discussion

The DC magnetic flux is plotted in Figure 5. The mutual inductance is evaluated using the line integral approach of the magnetic flux, as well as the point integral of the magnetic vector potential. These are both M = 1.978 nH and agree well with the analytic solution of 1.974 nH. In the limit as $R_1 >> R_2 >> r_0$, and with increasing mesh refinement, the numerical solution converges to the analytic one.

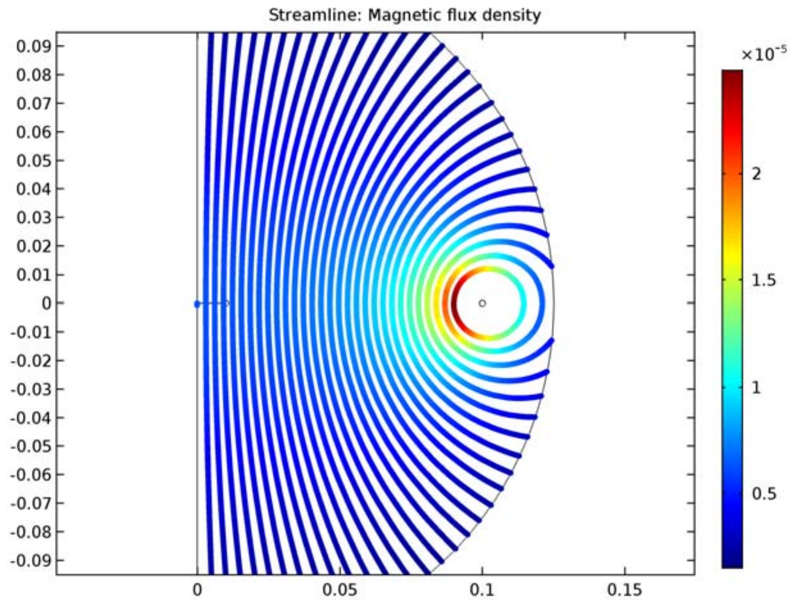


Figure 5: Magnetic flux lines for the DC case.

The induced currents of the secondary coil is plotted in Figure 6 for the open circuit case. The average of the induced currents over the cross section is zero, that is, there is no net current flow through the coil. The mutual inductance can be computed in three ways.

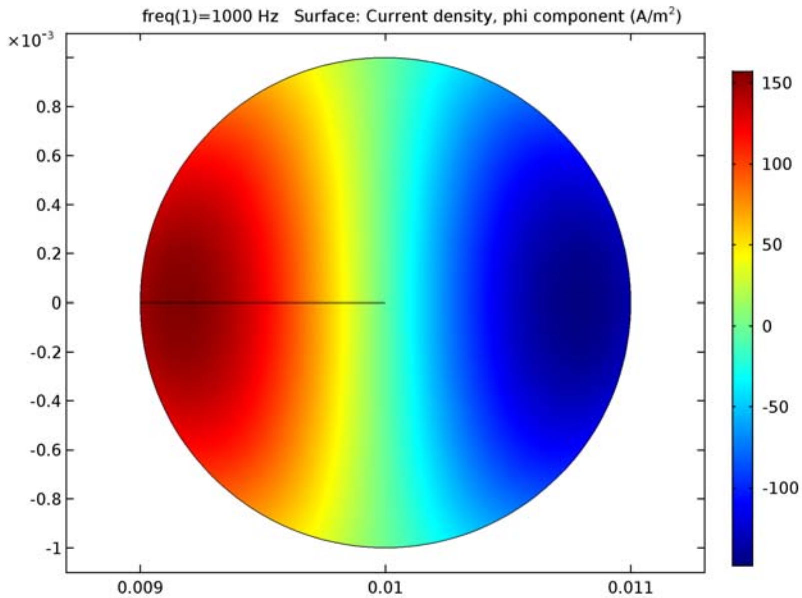


Figure 6: Induced currents in the coil for the open circuit case, the average of the current flow over the cross section is zero.

First, since the Coil feature computes the loop potential, this voltage induced across the coil can be used to compute the mutual inductance via:

$$M = \frac{V}{i\omega I} \tag{1}$$

The computed mutual inductance is 1.973 – 0.004i nH. The small imaginary component is due to the resistive effects, that is due to finite conductivity there are eddy current losses in the wires and the coil AC impedance (V/I), though mainly reactive, has a small resistive part.

The line integral of the magnetic flux can also be used, as in the DC case, and gives M=1.978-0.005i. Finally, by evaluating the integral of the magnetic vector potential over the entire cross section of the coil, and dividing by the area, M = 1.982 – 0.005i. There is a small difference between these three approaches, but they all agree well.

The induced currents of the secondary coil is plotted in Figure 7 for the closed circuit case. The skin effect is clearly visible; the current is being driven to the boundaries of the domain. The total induced current around the secondary coil is – 0.01675 – 0.02677i, the imaginary component implies a reactive current.

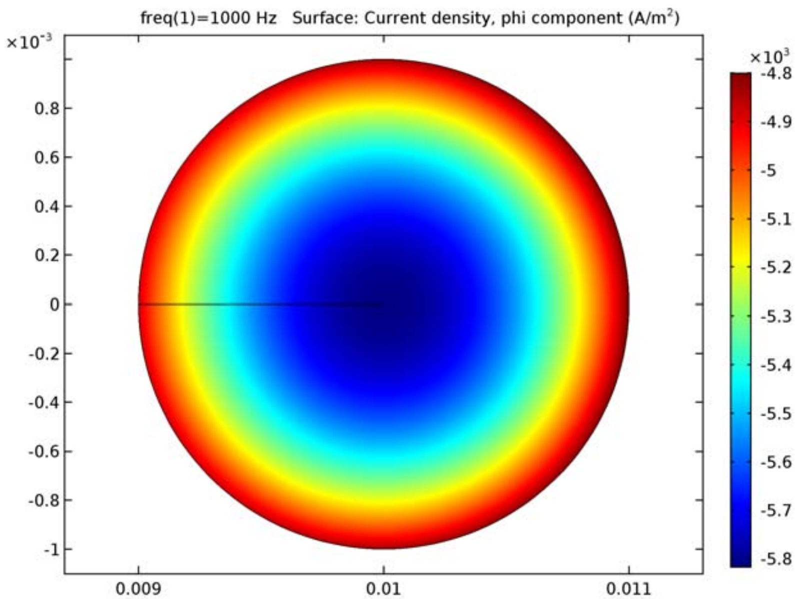


Figure 7: Induced currents in the coil for the closed circuit case.

Application Library path: `ACDC_Module/Inductive_Devices_and_Coils/mutual_inductance`

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 **2D Axisymmetric**.

2 In the **Select Physics** tree, select **AC/DC>Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
r_wire	1[mm]	0.001 m	Radius, wire
R1	100[mm]	0.1 m	Radius, outer coil
R2	10[mm]	0.01 m	Radius, inner coil
M	(mu0_const*pi* R2^2)/(2*R1)	1.9739E-9 H	Analytic mutual inductance

Here, `mu0_const` a predefined COMSOL constant for the permeability in vacuum.

GEOMETRY 1

Create a circle for the simulation domain. Define a layer in the circle where you will assign the Infinite Element Domain.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Sector angle** text field, type `180`.

4 In the **Radius** text field, type `1.75*R1`.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	50[mm]

Create a circle for the outer coil.

Circle 2 (c2)

1. On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
2. In the **Settings** window for Circle, locate the **Size and Shape** section.
3. In the **Radius** text field, type `r_wire`.
4. Locate the **Position** section. In the **r** text field, type `R1`.

   Then, create a circle for the inner coil.

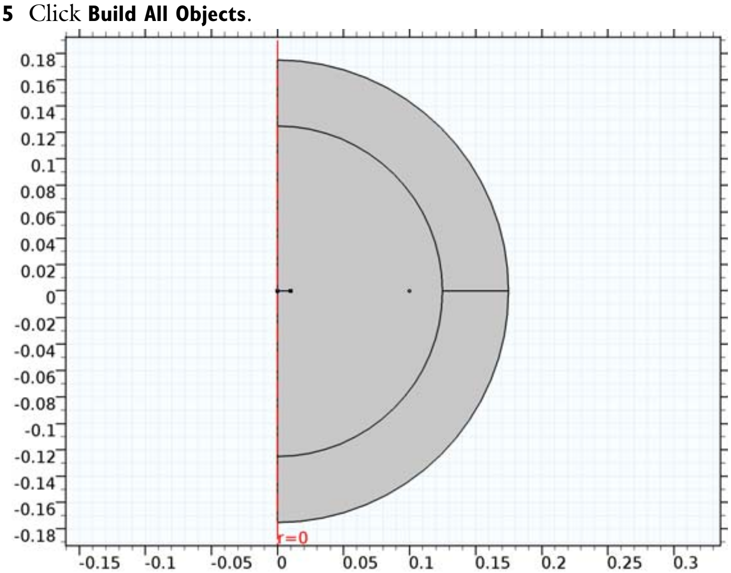
Circle 3 (c3)

1. On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
2. In the **Settings** window for Circle, locate the **Size and Shape** section.
3. In the **Radius** text field, type `r_wire`.
4. Locate the **Position** section. In the **r** text field, type `R2`.

   Add a line for computing the mutual inductance as a line integral of the magnetic flux.

Polygon 1 (pol1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Polygon**.
2. In the **Settings** window for Polygon, locate the **Coordinates** section.
3. In the **r** text field, type `0 R2`.
4. In the **z** text field, type `0 0`.
5. Click **Build All Objects**.



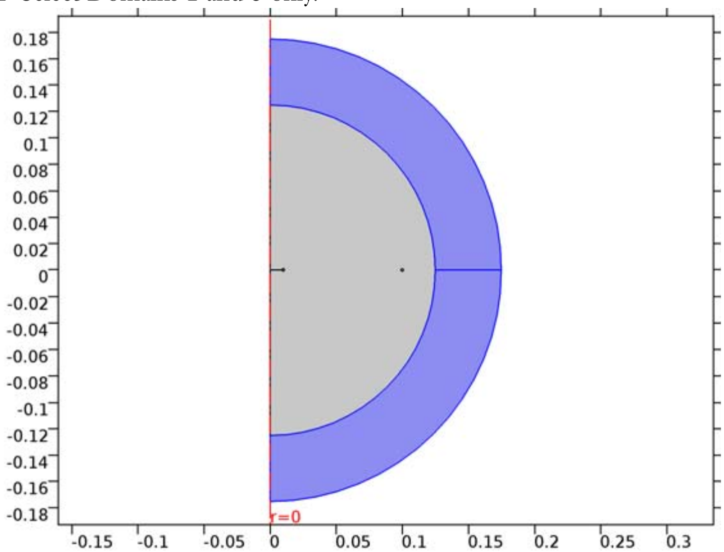
DEFINITIONS

Define the Infinite Element Domain to apply a coordinate transformation that mathematically stretches the layer to infinity.

Infinite Element Domain 1 (ie1)

1 On the **Definitions** toolbar, click **Infinite Element Domain**.

2 Select Domains 1 and 3 only.



MAGNETIC FIELDS (MF)

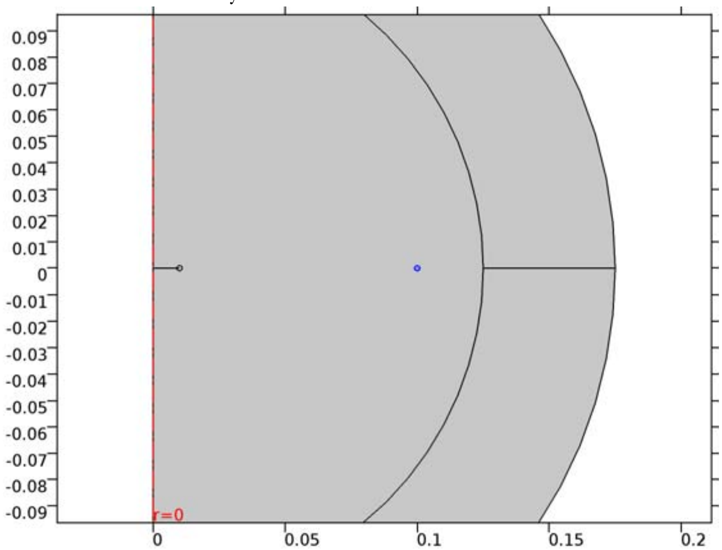
Now, set up the physics. Assign a **Coil** feature on the outer and the inner coil.

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

Coil 1

1 On the **Physics** toolbar, click **Domains** and choose **Coil**.

2 Select Domain 5 only.

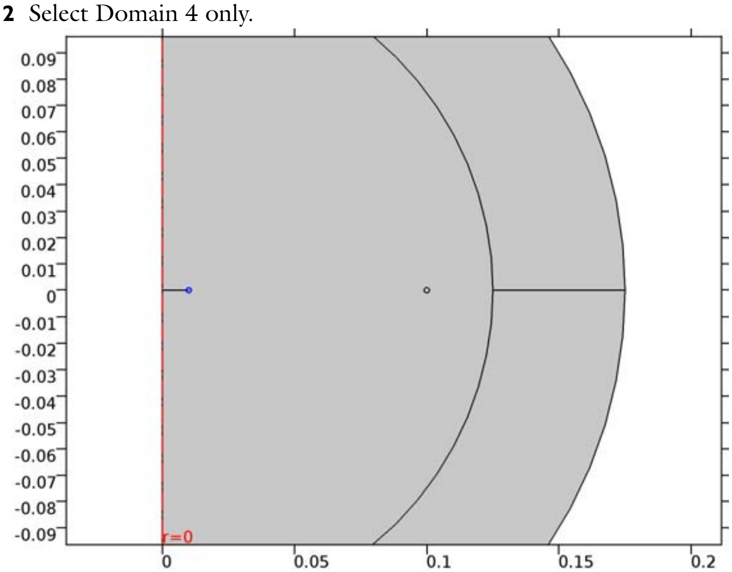


Specify `0[A]` current for the **Coil** feature assigned to the inner coil to model the open circuit case.

Coil 2

1 On the **Physics** toolbar, click **Domains** and choose **Coil**.

2 Select Domain 4 only.



3 In the **Settings** window for Coil, locate the **Coil** section.

4 In the I_{coil} text field, type `0[A]`.

MATERIALS

Next, assign material properties. Use **Air** for all domains.

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

MATERIALS

Air (mat1)

Then, override the coil domains with copper.

ADD MATERIAL

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

2 In the tree, select **AC/DC>Copper**.

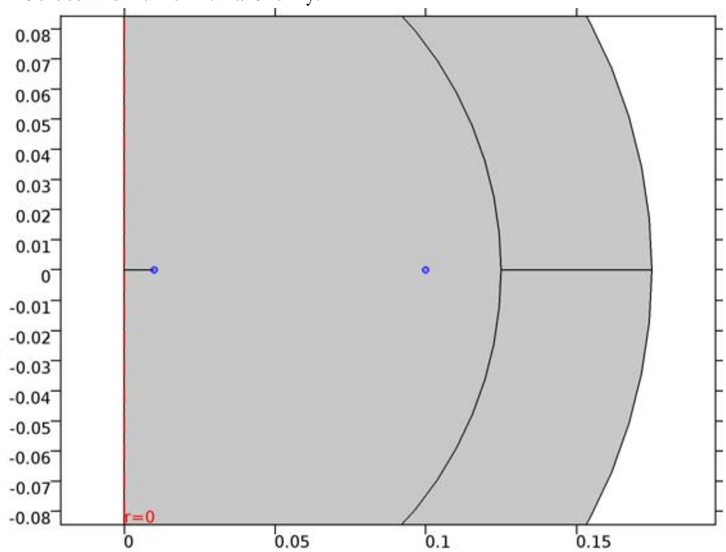
3 Click **Add to Component** in the window toolbar.

MATERIALS

Copper (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat2)**.

2 Select Domains 4 and 5 only.

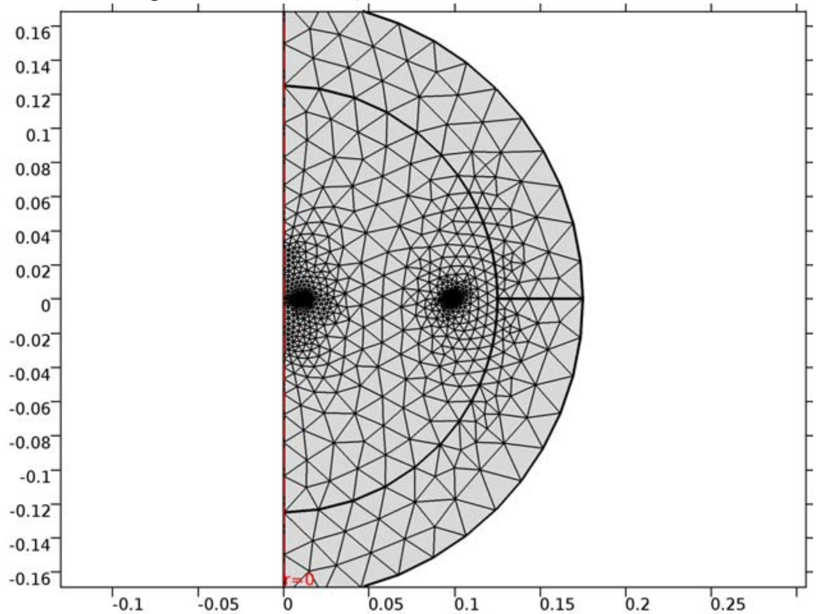


MESH 1

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

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

3 In the **Settings** window for Mesh, click **Build All**.



STUDY 1

1. In the **Settings** window for Study, locate the **Study Settings** section.
2. Clear the **Generate default plots** check box.
3. On the **Home** toolbar, click **Compute**.

RESULTS

In the **Model Builder** window, expand the **Results** node.

Study 1/Solution 1 (sol1)

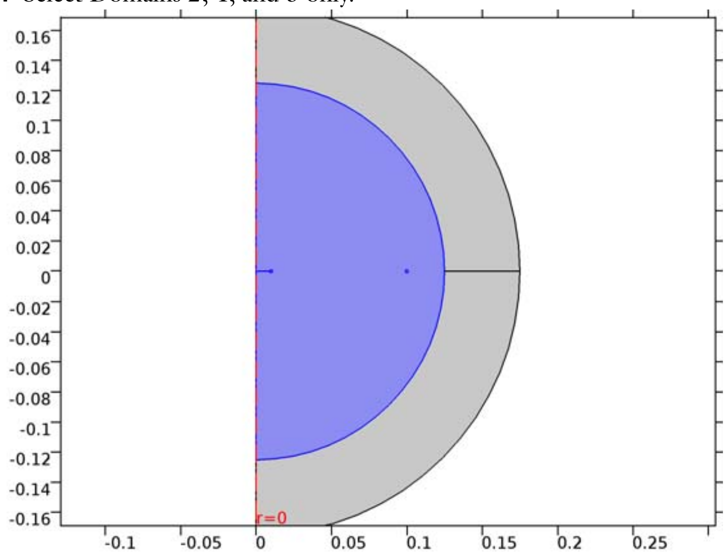
Select only the domains not part of the Infinite Element Domain for better magnetic flux visualization.

1. In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Solution 1 (sol1)**.

Selection

1. On the **Results** toolbar, click **Selection**.
2. In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2, 4, and 5 only.



2D Plot Group 1

On the **Results** toolbar, click **2D Plot Group**.

Streamline 1

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

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

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

4 From the **Entry method** list, choose **Coordinates**.

5 In the **r** text field, type `range(0,0.9*R1/29,0.9*R1)`.

6 In the **z** text field, type `0`.

7 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

Color Expression 1

1 Right-click **Results>2D Plot Group 1>Streamline 1** and choose **Color Expression**.

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

The resulting plot shows the magnetic flux lines for the DC case as in Figure 5.

Derived Values

Evaluate the mutual inductance using the line integral of the magnetic flux density.

Line Integration 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Line Integration**.
2 Click the **Zoom Extents** button on the **Graphics** toolbar.
3 Select Boundaries 4 and 6 only.
4 In the **Settings** window for Line Integration, locate the **Expressions** section.
5 In the table, enter the following settings:

Expression	Unit	Description
`mf.Bz/1[A]`	H	

6 Locate the **Integration Settings** section. Select the **Compute surface integral** check box.
7 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`mf.Bz/1[A]`	nH	

8 Click **Evaluate**.

Derived Values

Evaluate the mutual inductance using the point integral of the magnetic vector potential.

Point Evaluation 1

1 On the **Results** toolbar, click **Point Evaluation**.
2 Select Point 8 only.
3 In the **Settings** window for Point Evaluation, locate the **Expressions** section.
4 In the table, enter the following settings:

Expression	Unit	Description
`2*pi*r*Aphi/1[A]`	nH	

5 Click **Evaluate**.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2 Go to the **Add Study** window.
3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Frequency Domain**.
4 Click **Add Study** in the window toolbar.

STUDY 2

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2** 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 `1[kHz]`.

4 In the **Model Builder** window, click **Study 2**.

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

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

7 On the **Home** toolbar, click **Compute**.

RESULTS

Study 2/Solution 2 (sol2)

Select the inner coil domain.

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 2/ Solution 2 (sol2)**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 Select Domain 4 only.

2D Plot Group 2

1 On the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for 2D Plot Group, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

Surface 1

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

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields> Currents and charge>Current density>mf.Jphi - Current density, phi component**.

3 On the **2D Plot Group 2** toolbar, click **Plot**.

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

Compare the reproduced plot with Figure 6.

Derived Values

Evaluate the mutual inductance using Equation 1.

Global Evaluation 1

1. On the **Results** toolbar, click **Global Evaluation**.
2. In the **Settings** window for Global Evaluation, locate the **Data** section.
3. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.
4. Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`mf.VCoil_2/1[A]/mf.iomega`	nH	

5. Click **Evaluate**.

Derived Values

Evaluate the mutual inductance using the line integral of the magnetic flux density.

Line Integration 2

1. On the **Results** toolbar, click **More Derived Values** and choose **Integration>Line Integration**.
2. In the **Settings** window for Line Integration, locate the **Data** section.
3. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.
4. Select Boundaries 4 and 6 only.
5. Locate the **Integration Settings** section. Select the **Compute surface integral** check box.
6. Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`mf.Bz/1[A]`	nH	

7. Click **Evaluate**.

Derived Values

Evaluate the mutual inductance using the integral of the magnetic vector potential over the entire cross section of the coil.

Surface Integration 3

1. On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.
2. In the **Settings** window for Surface Integration, locate the **Data** section.
3. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 Select Domain 4 only.

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

Expression	Unit	Description
`(2*pi*r*Aphi)/(pi*r_wire^2)/1[A]`	`nH`	

6 Click **Evaluate**.

Specify `0[V]` voltage for the **Coil** feature assigned to the inner coil to model the closed circuit case.

MAGNETIC FIELDS (MF)

Coil 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Fields (mf)** click **Coil 2**.

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

3 From the **Coil excitation** list, choose **Voltage**.

4 In the V_{coil} text field, type `0`.

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Frequency Domain**.

3 Click **Add Study** in the window toolbar.

STUDY 3

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 3** 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 `1[kHz]`.

4 In the **Model Builder** window, click **Study 3**.

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

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

7 On the **Home** toolbar, click **Compute**.

RESULTS

Study 3/Solution 3 (sol3)

Select the inner coil domain.

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 3/ Solution 3 (sol3)**.

Selection

1 On the **Results** toolbar, click **Selection**.
2 In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domain 4 only.

2D Plot Group 3

1 On the **Results** toolbar, click **2D Plot Group**.
2 In the **Settings** window for 2D Plot Group, locate the **Data** section.
3 From the **Data set** list, choose **Study 3/Solution 3 (sol3)**.

Surface 1

1 Right-click **2D Plot Group 3** and choose **Surface**.
2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields> Currents and charge>Current density>mf.Jphi - Current density, phi component**.
3 On the **2D Plot Group 3** toolbar, click **Plot**.
4 Click the **Zoom Extents** button on the **Graphics** toolbar.

The reproduced plot should look like Figure 7.

Derived Values

Evaluate the total induced current on the inner (secondary) coil.

Global Evaluation 2

1 On the **Results** toolbar, click **Global Evaluation**.
2 In the **Settings** window for Global Evaluation, locate the **Data** section.
3 From the **Data set** list, choose **Study 3/Solution 3 (sol3)**.
4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
mf.ICoil_2	A	Coil current

5 Click **Evaluate**.

Created in COMSOL Multiphysics 5.2a

Electromagnetic Forces on Parallel Current-Carrying Wires

Introduction

One ampere is defined as the constant current in two straight parallel conductors of infinite length and negligible circular cross section, placed one meter apart in vacuum, that produces a force of 2×10^{-7} newton per meter of length (N/m). This example shows a setup of two parallel wires in the spirit of this definition, but with the difference that the wires have finite cross sections.

For wires with circular cross section carrying a uniform current density as in this example, the mutual magnetic force is the same as for line currents. This can be understood by the following arguments: Start from a situation where both wires are line currents ($\mathbf{I}$). Each line current is subject to a Lorentz force ($\mathbf{I} \times \mathbf{B}$), where the magnetic flux density ($\mathbf{B}$) is the one produced by the other wire. Now, give one wire a finite radius. It follows directly from circular symmetry and Maxwell-Ampère's law that, outside this wire, the produced flux density is exactly the same as before so the force on the remaining line current is unaltered. Further, the net force on the wire with the distributed current density must be of exactly the same magnitude (but with opposite direction) as the force on the line current so that force did not change either. If the two wires exchange places, the forces must still be the same, and it follows from symmetry that the force is independent of wire radius as long as the wire cross sections do not intersect. The wires can even be cylindrical shells or any other shape with circular symmetry. For an experimental setup, negligible cross section is required as resistive voltage drop along the wires and Hall effect may cause electrostatic forces that increase with wire radius but such effects are not included in this example.

The force between the wires is computed using two different methods: first automatically by integrating the stress tensor on the boundaries, then by integrating the volume (Lorentz) force density over the wire cross section. The results converge to 2×10^{-7} N/m for the 1 ampere definition, as expected.

Model Definition

The application is built using the 2D Magnetic Fields interface. The modeling plane is a cross section of the two wires and the surrounding air.

DOMAIN EQUATIONS

The equation formulation assumes that the only nonzero component of the magnetic vector potential is A_z. This corresponds to all currents being perpendicular to the modeling plane. The following equation is solved:

$$\nabla \times (\mu \nabla \times A_z) = J_z^{\mathrm{e}}$$

where μ is the permeability of the medium and J_z^{e} is the externally applied current. J_z^{e} is set so that the applied current in the wires equals 1 A, but with different signs.

Surrounding the air is an infinite element domain. For details, see the *AC/DC Module User's Guide*.

Results and Discussion

The expression for the surface stress reads

$$\mathbf{n}_1 T_2 = -\frac{1}{2}(\mathbf{H} \cdot \mathbf{B})\mathbf{n}_1 + (\mathbf{n}_1 \cdot \mathbf{H})\mathbf{B}^{\mathrm{T}}$$

where $\mathbf{n}_1$ is the boundary normal pointing out from the conductor wire and T_2 the stress tensor of air. The closed line integral of this expression around the circumference of either wire evaluates to -1.99×10^{-7} N/m. The minus sign indicates that the force between the wires is repulsive. The software automatically provides the coordinate components of the force on each wire.

The volume force density is given by

$$\mathbf{F} = \mathbf{J} \times \mathbf{B} = \left[-J_z^{\mathrm{e}} \cdot B_y, J_z^{\mathrm{e}} \cdot B_x, 0\right]$$

The surface integral of the x component of the volume force on the cross section of a wire gives the result -2.00×10^{-7} N/m.

By refining the mesh and re-solving the problem, you can verify that the solution with both method converges to -2×10^{-7} (N/m), see Mesh Convergence. The volume force density integral is typically the most accurate one for reasons explained in the *COMSOL Multiphysics Reference Manual*.

Mesh Convergence

In order to investigate the accuracy of the model, it is recommended to perform a systematic mesh convergence analysis of the desired entity, here the force on the wire. In Figure 1 and Figure 2, the mesh convergence is shown for the absolute errors in the Maxwell surface stress method and the volumetric Lorentz force method, respectively. The

Lorentz force is 2–3 orders of magnitude more accurate than the Maxwell stress tensor force for a given mesh density.

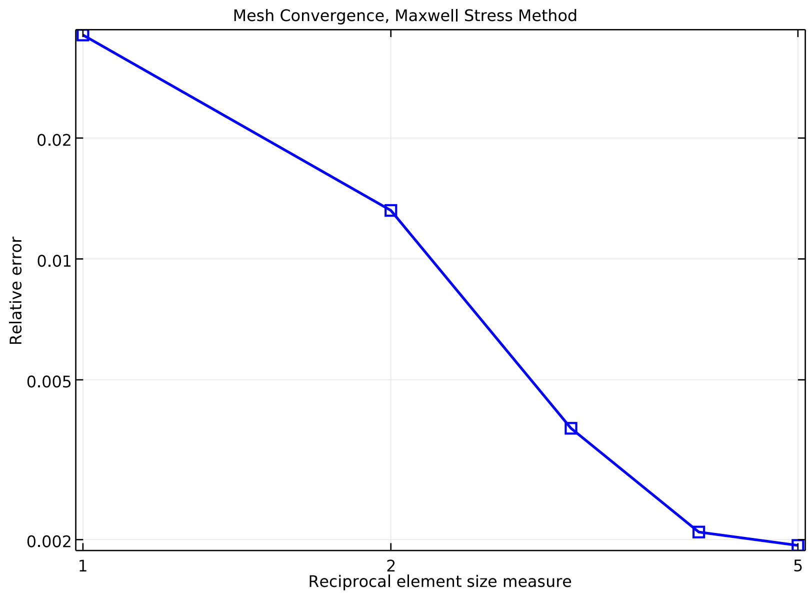


Figure 1: Mesh convergence is shown for the force computation using the Maxwell surface stress method.

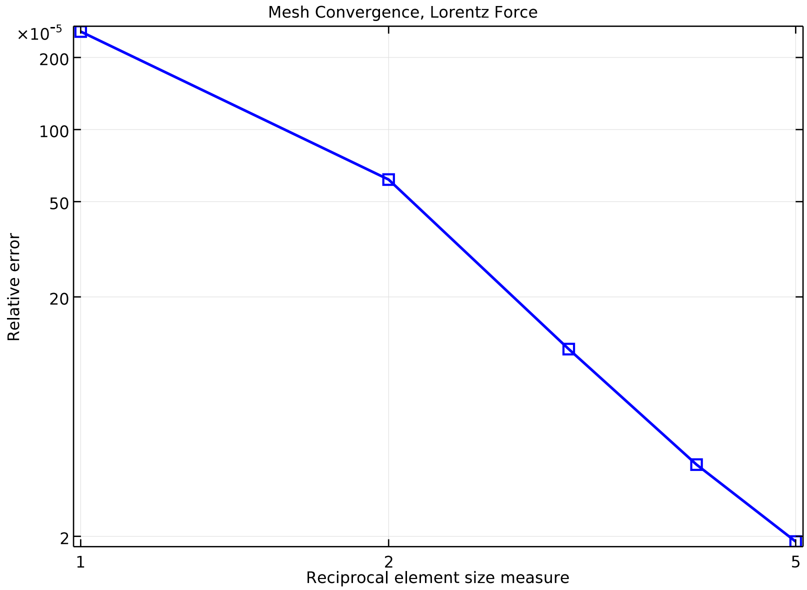


Figure 2: Mesh convergence is shown for the force computation using the volumetric Lorentz force method.

Application Library path: `ACDC_Module/Verification_Examples/parallel_wires`

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 **2D**.

2 In the **Select Physics** tree, select **AC/DC>Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
r	0.2[m]	0.2 m	Wire radius
I0	1[A]	1 A	Total current
J0	I0/(pi*r^2)	7.9577 A/m²	Current density
N	1	1	Mesh multiplier

GEOMETRY 1

Add a circle for the main air domain. The outer layer will constitute an infinite element domain to approximate a region extending to infinity.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `1.5`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.5

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `r`.

4 Locate the **Position** section. In the **x** text field, type `0.5`.

Circle 3 (c3)

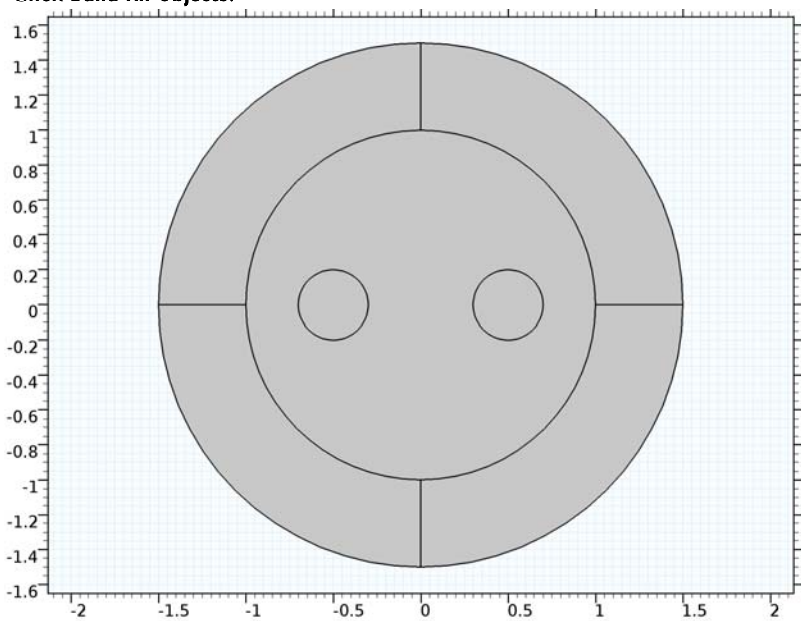
1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `r`.

4 Locate the **Position** section. In the **x** text field, type `-0.5`.

5 Click **Build All Objects**.



You have now drawn the wires in their surrounding air domain plus an infinite element layer.

Define an infinite element region in the outer domains.

DEFINITIONS

Infinite Element Domain 1 (ie1)

1 On the **Definitions** toolbar, click **Infinite Element Domain**.

2 Select Domains 1–4 only.

3 In the **Settings** window for Infinite Element Domain, locate the **Geometry** section.

4 From the **Type** list, choose **Cylindrical**.

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat1)

By default, the first material you select will apply to your entire geometry. Air is defined with a zero conductivity, and relative permittivity and permeability both equal to 1. These properties are the same as those of vacuum, which is the assumed material in the Ampère definition. Since the model assumes a given uniform current distribution, it is safe to use the same properties in the wires too.

MAGNETIC FIELDS (MF)

External Current Density 1

1 On the **Physics** toolbar, click **Domains** and choose **External Current Density**.

2 Select Domain 6 only.

3 In the **Settings** window for External Current Density, locate the **External Current Density** section.

4 Specify the $\mathbf{J}_e$ vector as

0	x
0	y
J0	z

External Current Density 2

1 On the **Physics** toolbar, click **Domains** and choose **External Current Density**.

2 Select Domain 7 only.

3 In the **Settings** window for External Current Density, locate the **External Current Density** section.

4 Specify the $\mathbf{J}_e$ vector as

0	x
0	y
-J0	z

You have now completely defined the physics of this model. Adding Force Calculation features will make the Maxwell's stress tensors available as a variable.

Force Calculation 1

1 On the **Physics** toolbar, click **Domains** and choose **Force Calculation**.
2 Select Domain 6 only.
3 In the **Settings** window for Force Calculation, locate the **Force Calculation** section.
4 In the **Force name** text field, type `wire1`.

Force Calculation 2

1 On the **Physics** toolbar, click **Domains** and choose **Force Calculation**.
2 Select Domain 7 only.
3 In the **Settings** window for Force Calculation, locate the **Force Calculation** section.
4 In the **Force name** text field, type `wire2`.

MESH 1

The infinite element domain requires some extra attention when meshing. As it is steeply scaled in the radial direction to make the geometry effectively extend to infinity, the mesh will effectively be stretched in that direction. To avoid poor element quality, a structured mesh is used.

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.
2 In the **Settings** window for Mapped, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domains 1–4 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.
2 Select Boundaries 5–8 and 13–16 only.
3 In the **Settings** window for Distribution, locate the **Distribution** section.
4 In the **Number of elements** text field, type `10`.

Mapped 1

Right-click **Mapped 1** and choose **Distribution**.

Distribution 2

1 Select Boundaries 1–4 only.
2 In the **Settings** window for Distribution, locate the **Distribution** section.

3 In the **Number of elements** text field, type `4`.

Size

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

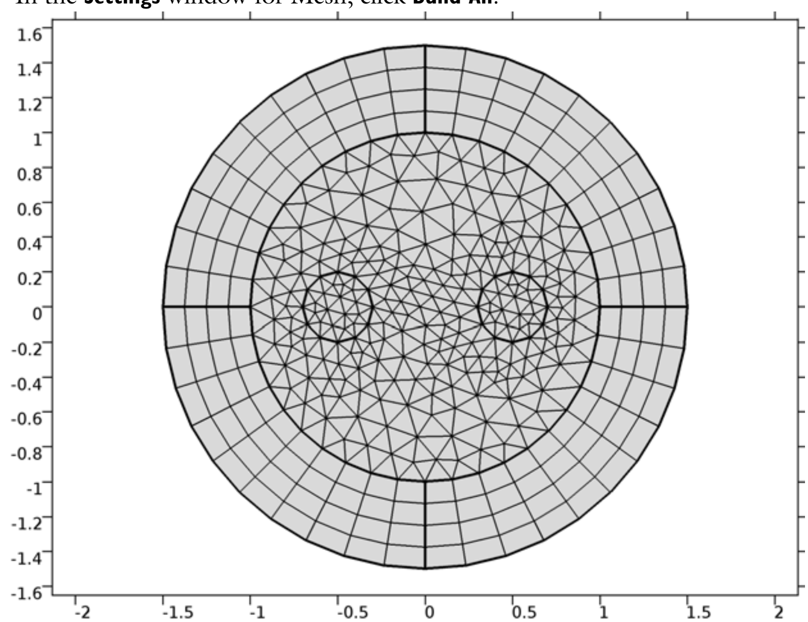
2 In the **Settings** window for Size, locate the **Element Size** section.

3 From the **Predefined** list, choose **Coarser**.

4 Click to expand the **Element size parameters** section. Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.2`.

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

6 In the **Settings** window for Mesh, click **Build All**.



The mesh should look like in the figure.

STUDY 1

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

RESULTS

Magnetic Flux Density Norm (mf)

The default plot shows the norm of the magnetic flux density. Note that the value inside the infinite element domain has no physical relevance.

Arrow Line 1

1 In the **Model Builder** window, under **Results** right-click **Magnetic Flux Density Norm (mf)** and choose **Arrow Line**.

2 In the **Settings** window for Arrow Line, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields>Mechanical>mf.nTx_wire1,mf.nTy_wire1 - Maxwell surface stress tensor**.

3 Locate the **Coloring and Style** section. Select the **Scale factor** check box.

4 In the associated text field, type 300000.

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

6 On the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

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

Magnetic Flux Density Norm (mf)

Right-click **Magnetic Flux Density Norm (mf)** and choose **Arrow Line**.

Arrow Line 2

1 In the **Settings** window for Arrow Line, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields>Mechanical>mf.nTx_wire2,mf.nTy_wire2 - Maxwell surface stress tensor**.

2 Locate the **Coloring and Style** section. Select the **Scale factor** check box.

3 In the associated text field, type 300000.

4 From the **Color** list, choose **Black**.

5 On the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

You are now looking at the Maxwell's stress tensor distribution on the surface of the wires. The total force on each wired is evaluated as the surface integral of the stress tensor, and stored in an automatically available expression.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for Global Evaluation, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Magnetic Fields>Mechanical>Electromagnetic force>mf.Forcex_wire1 - Electromagnetic force, x component**.

3 Click **Evaluate**.

The force in the x-direction on the first wire will evaluate to something between -2.0×10^{-7} N/m and -1.9×10^{-7} N/m.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Magnetic Fields>Mechanical>Electromagnetic force> mf.Forcex_wire2 - Electromagnetic force, x component**.

5 Click **Evaluate**.

As expected, the force on the second wire has a similar value but the opposite sign.

Proceed to compare the values you just got with those from the Lorentz force distribution.

Surface Integration 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.

2 Select Domain 6 only.

3 In the **Settings** window for Surface Integration, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model> Component 1>Magnetic Fields>Mechanical>Lorentz force contribution>mf.FLtzx - Lorentz force contribution, x component**.

4 Click **Evaluate**.

This time, the value is expected to be consistently closer to -2×10^{-7} N/m. When applicable, Lorentz force integrals usually give more accurate results than the Maxwell's stress tensor.

5 Select Domain 7 only.

6 Click **Evaluate**.

Once again, integration over the second wire gives a similar but positive result.

MESH 1

Proceed with the mesh convergence analysis for the force. First a parameterized mesh is created.

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Duplicate**.

MESH 2

Size

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Meshes>Mesh 2** node, then click **Size**.

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

3 In the **Maximum element size** text field, type `0.2/N`.

Distribution 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Meshes>Mesh 2>Mapped 1** node, then click **Distribution 1**.

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

3 In the **Number of elements** text field, type `10*N`.

Distribution 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes>Mesh 2>Mapped 1** click **Distribution 2**.

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

3 In the **Number of elements** text field, type `4*N`.

ROOT

Perform the mesh convergence analysis in a new study.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Perform a sweep over the mesh multiplier parameter.

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
N		

5 Click **Range**.

6 In the **Range** dialog box, type `1` in the **Start** text field.

7 In the **Step** text field, type `1`.

8 In the **Stop** text field, type `5`.

9 Click **Replace**.

DEFINITIONS

Define an integration operator for the plotting of the Lorentz force.

Integration 1a (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 Select Domain 7 only.

STUDY 2

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

RESULTS

1D Plot Group 3

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

Plot the absolute error versus the mesh multiplier parameter for the Maxwell stress tensor based force.

2 In the **Settings** window for 1D Plot Group, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Parametric Solutions 1 (sol3)**.

Global 1

1 On the **1D Plot Group 3** toolbar, click **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
`abs(mf.Forcex_wire2-2e-7)/2e-7`	N	

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

6 From the **Positioning** list, choose **In data points**.

7 On the **1D Plot Group 3** toolbar, click **Plot**.

Switch to logarithmic scale and add suitable plot annotations.

8 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

9 Click the **y-Axis Log Scale** button on the **Graphics** toolbar.

1D Plot Group 3

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

2 In the **Settings** window for 1D Plot Group, click to expand the **Title** section.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Mesh Convergence, Maxwell Stress Method`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `Reciprocal element size measure`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Relative error`.

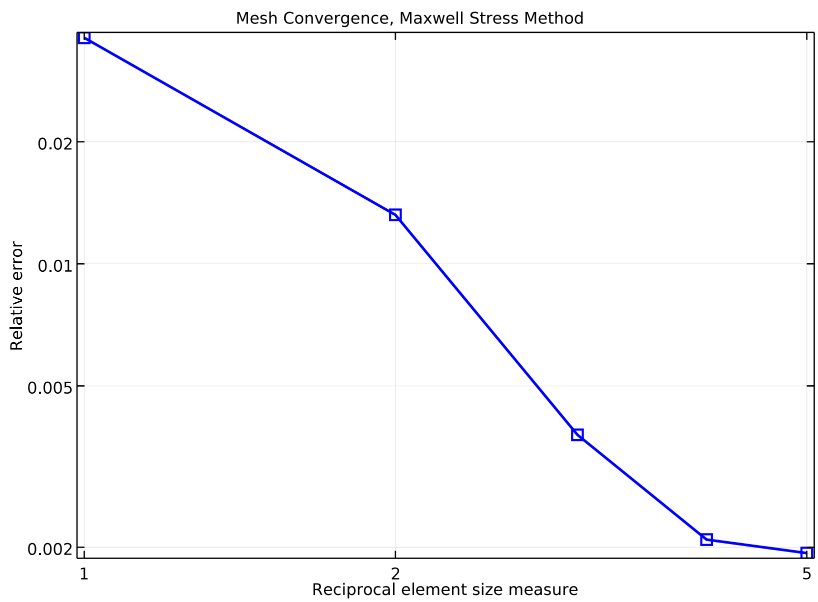
Global 1

1 In the **Model Builder** window, under **Results>1D Plot Group 3** click **Global 1**.

2 In the **Settings** window for Global, click to expand the **Legends** section.

3 Clear the **Show legends** check box.

4 On the **1D Plot Group 3** toolbar, click **Plot**.



1D Plot Group 3

Plot the absolute error versus the mesh multiplier parameter for the Lorentz force.

1D Plot Group 4

1. In the **Model Builder** window, under **Results** right-click **1D Plot Group 3** and choose **Duplicate**.
2. In the **Settings** window for 1D Plot Group, locate the **Title** section.
3. In the **Title** text area, type `Mesh Convergence, Lorentz Force`.

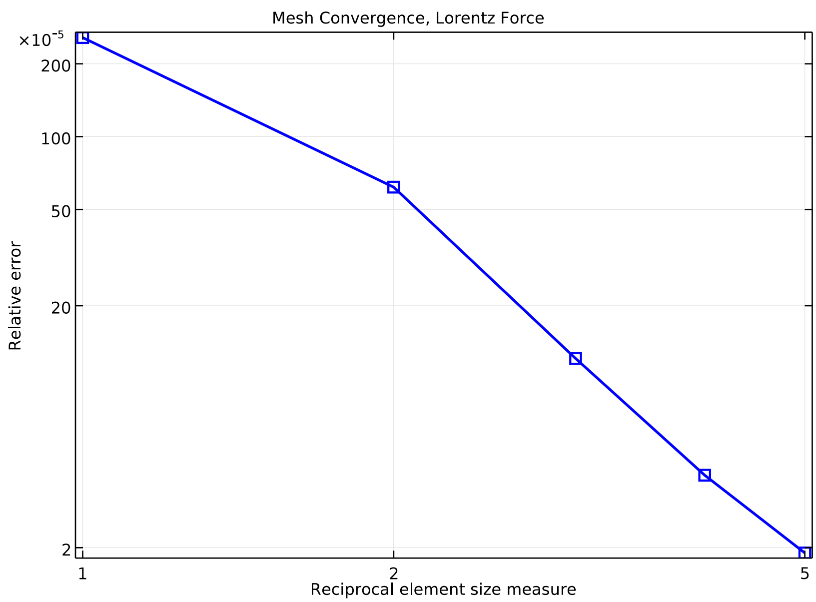
Global 1

1. In the **Model Builder** window, expand the **1D Plot Group 4** node, then click **Global 1**.
2. In the **Settings** window for Global, locate the **y-Axis Data** section.
3. In the table, enter the following settings:

Expression	Unit	Description
`abs(intop1(mf.FLtzx)-2e-7)/2e-7`	`N/m`	

1D Plot Group 4

1. In the **Model Builder** window, under **Results** click **1D Plot Group 4**.
2. On the **1D Plot Group 4** toolbar, click **Plot**.



The Lorentz force is 2-3 orders of magnitude more accurate than the Maxwell stress tensor force for a given mesh density.

Magnetically Permeable Sphere in a Static Magnetic Field

Introduction

A sphere of relative permeability greater than unity is exposed to a spatially uniform static background magnetic field. Two formulations are used to solve this problem, and the differences between these are discussed. The field strength inside the sphere is computed and compared against the analytic solution.

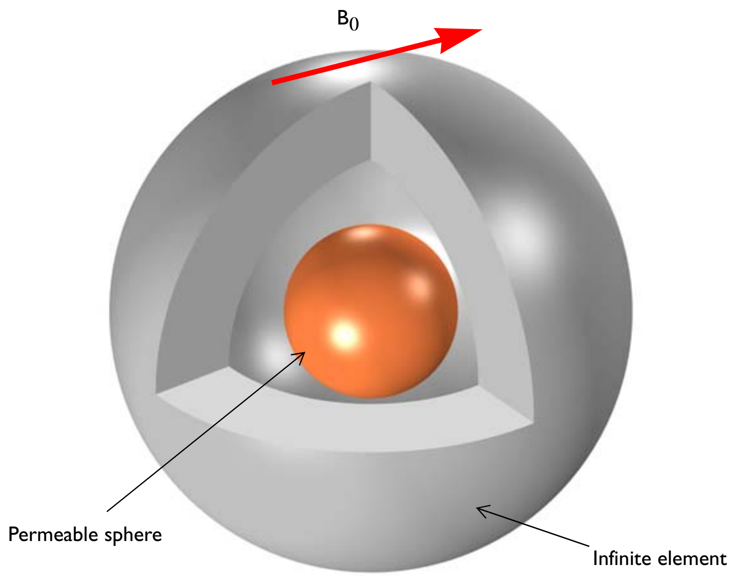


Figure 1: An magnetically permeable sphere in a spatially uniform, static background magnetic field. The sphere at the center is surrounded by air, and enclosed in a region of Infinite Elements.

Model Definition

Figure 1 shows the model setup, with a 0.25 mm diameter sphere placed in a spatially uniform background magnetic field of strength 1 mT. The computational model consists of three concentric spheres. The innermost is the permeable sphere, the surrounding spherical shell volume represents free space, and the outside shell volume represents a region extending to infinity, modeled with an Infinite Element Domain. When using Infinite Element Domain features, the boundary condition on the outside of the modeling domain does not affect the solution.

The relative permeability of the sphere is varied from $\mu_r = 2$ to $\mu_r = 1000$. The analytic solution for the field inside a permeable sphere exposed to a uniform magnetic field is:

$$\mathbf{B} = \mathbf{B}_0\left(\frac{3\mu_r}{\mu_r + 2}\right)$$

Where B_0 is the background magnetic field.

There are two ways in which this problem can be formulated. The scalar potential formulation, used in the Magnetic Fields, No Currents interface, solves the magnetic flux conservation equation:

$$\nabla \cdot \mathbf{B} = 0$$

a partial differential equation for the magnetic scalar potential field, V_m:

$$\nabla \cdot \mu_r\mu_0(-\nabla V_m + \mathbf{H}_b) = 0$$

where the background field is specified in terms of the **H**-field, $\mathbf{H}_b$. The **B**-field is then computed from the **H**-field: $\mathbf{B} = \mu_r\mu_0\mathbf{H}$. The magnetic field is in turn computed from the gradient of the magnetic scalar potential. Because the governing equation evaluates the gradients of a scalar field, the Lagrange element formulation is used. In this formulation, the background field and boundary conditions for this problem are specified purely in terms of derivatives of the V_m field, and the solution is unique up to a constant. To remove this indeterminacy, the value of the magnetic scalar potential must be constrained at one point in the model, to fix the value of the constant.

The vector potential formulation, used in the Magnetic Fields interface, solves an equation for the magnetic vector potential, **A**:

$$\nabla\times{\mu_r}^{-1}{\mu_0}^{-1}\nabla\times(\mathbf{A} + \mathbf{A}_b) = 0$$

Where the **B**-field is the curl of the $(\mathbf{A} + \mathbf{A}_b)$ field. In this approach, the background field and boundary conditions are specified directly in terms of the **A**-field. Here, the governing equation takes the curl of a vector valued field, and this problem is solved using a Curl element formulation. This formulation does not require as fine of a mesh as the Lagrange element formulation to achieve the same accuracy.

Results and Discussion

Figure 2 plots the magnetic field for the Magnetic Fields, No Currents interface formulation, and Figure 3 shows the results computed using the Magnetic Fields interface formulation, both for the $\mu_r = 1000$ case. The fields in the Infinite Element region are not plotted, as these do not have any physical significance.

Figure 4 shows the field enhancement versus the permeability for both cases, along with the analytic solution. The relative difference is plotted in Figure 5. In the limit as the mesh is refined the solutions agree to within numerical precision.

There are some differences between the two formulations. In this case, the Magnetic Fields interface slightly underestimates the field strength, while the Magnetic Fields, No Current interface overestimates it. The agreement with the analytic solution for both formulations improves with increasing mesh refinement. Although the Magnetic Fields, No Currents interface require a finer mesh for approximately the same level of accuracy, it does use less total memory. Its drawback is that it cannot be used to model situations where there is any current flowing in the model, or any variation with respect to time.

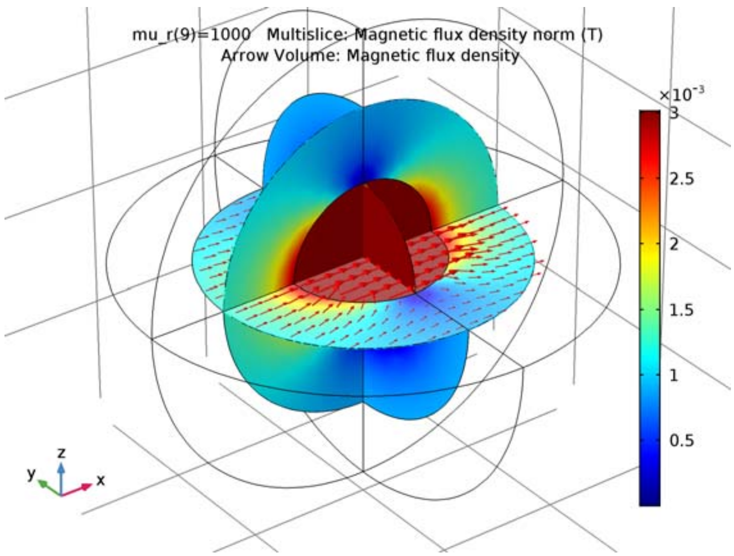


Figure 2: The magnetic field for the Magnetic Fields, No Currents interface.

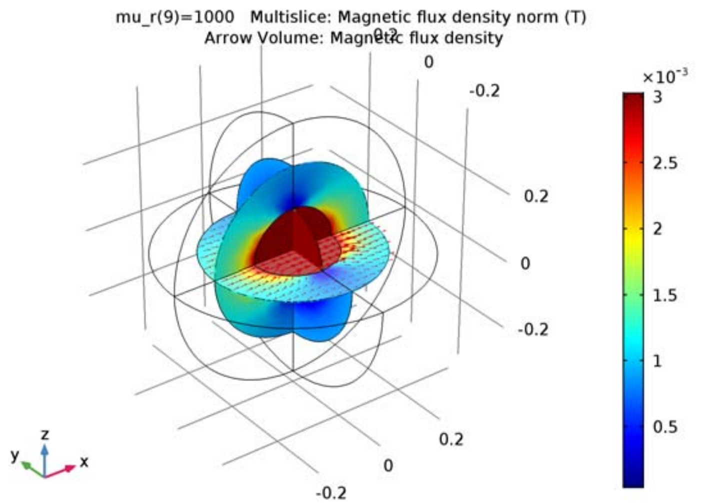


Figure 3: The magnetic field for the Magnetic Fields interface.

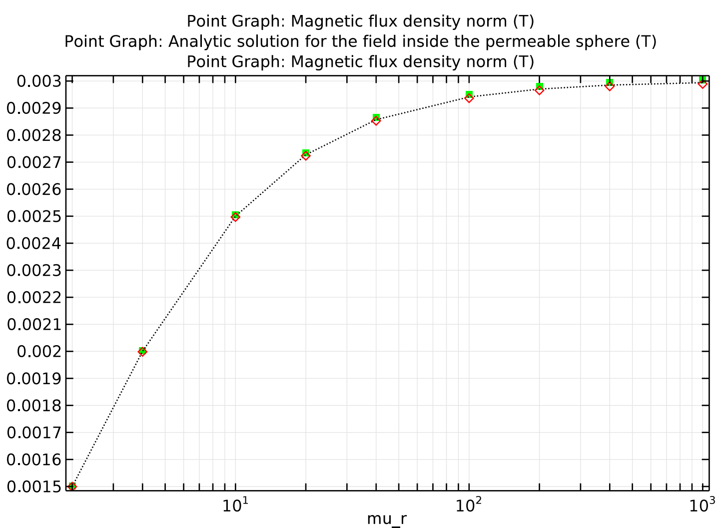


Figure 4: Comparison of numerical results to analytic result.

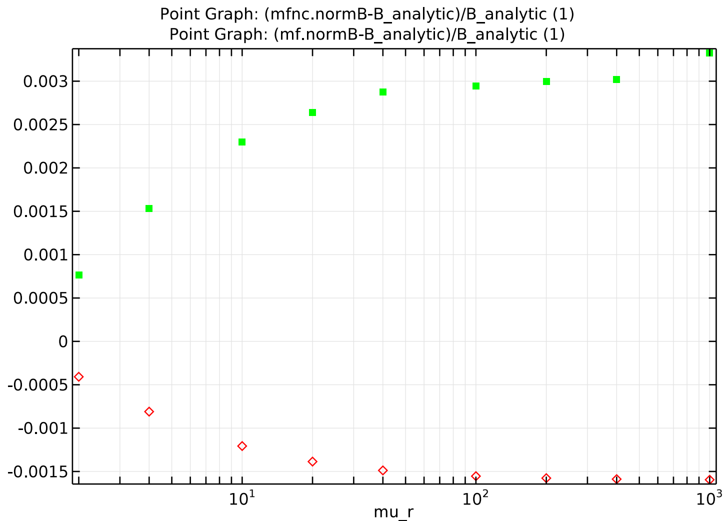


Figure 5: Relative difference compared to the analytic solution.

Application Library path: `ACDC_Module/Tutorials/permeable_sphere_static_bfield`

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>Magnetic Fields, No Currents (mfnc)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

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

2 In the **Settings** window for Geometry, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`r0`	`0.125[mm]`	1.25E-4 m	`Radius, magnetically permeable sphere`
`mu_r`	`1000`	1000	`Relative permeability, magnetically permeable sphere`
`B0`	`1[mT]`	0.001 T	`Background magnetic fields`
`B_analytic`	`((3*mu_r)/(mu_r+2))*B0`	0.002994 T	`Analytic solution for the field inside the permeable sphere`

GEOMETRY 1

Create a sphere with two layer definitions. The outermost layer represents the Infinite Element Domain and the core represents the permeable sphere. The median layer is the air domain.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for Sphere, locate the **Size** section.

3 In the **Radius** text field, type `r0*3`.

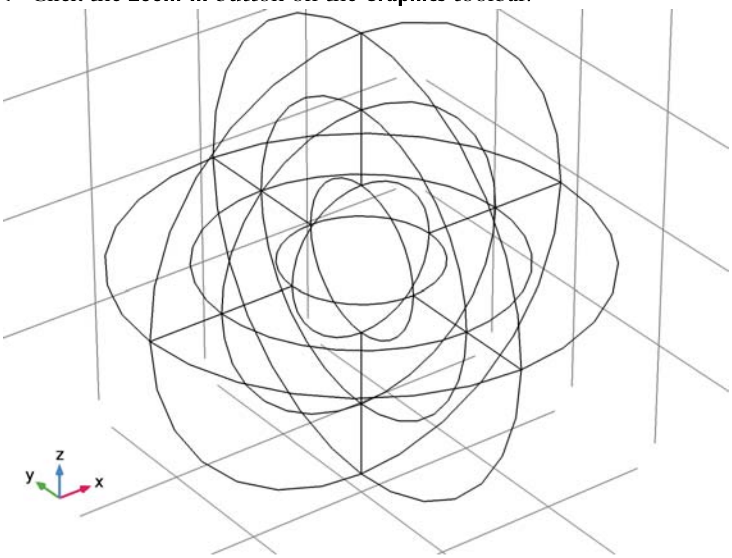
4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	`r0`
`Layer 2`	`r0`

5 Click **Build All Objects**.

6 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

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



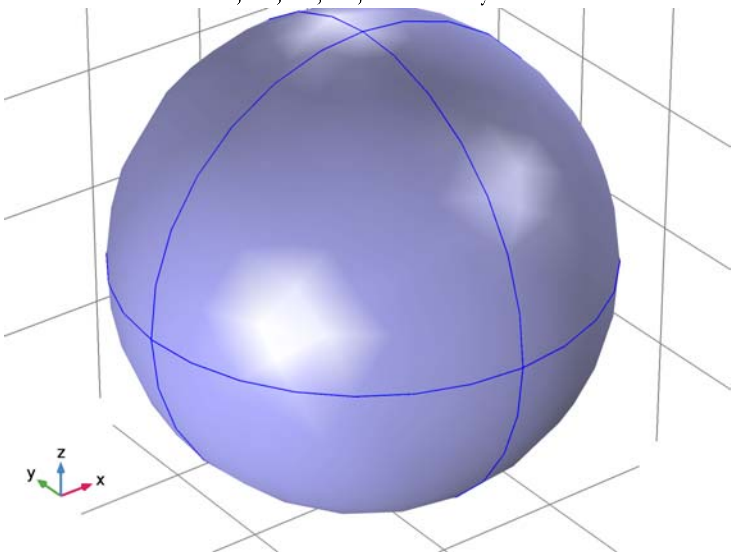
DEFINITIONS

Create a set of selections before setting up the physics. First, create a selection for the Infinite Element Domain.

Explicit 1

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

2 Select Domains 1–4, 10, 11, 14, and 17 only.



3 Right-click **Explicit 1** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Infinite Element domains` in the **New label** text field.

5 Click **OK**.

Add a selection for the analysis domains. This is the complement of the Infinite Element Domain selection.

Complement 1

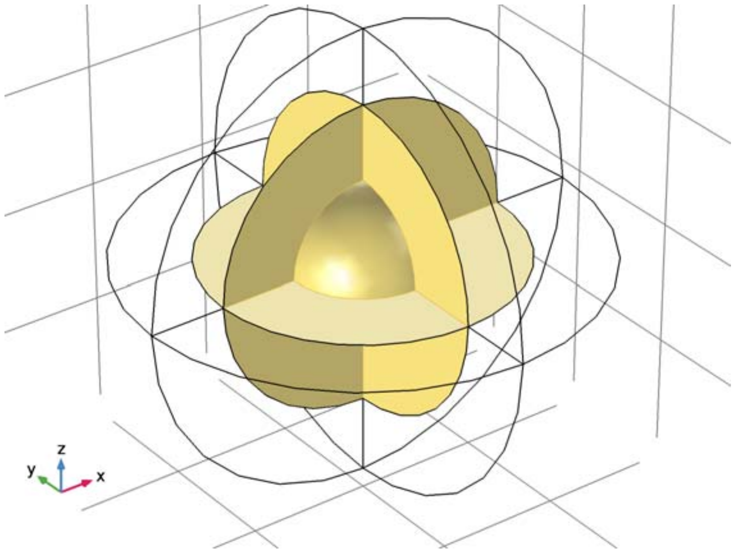
1 On the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for Complement, locate the **Input Entities** section.

3 Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **Infinite Element domains** in the **Selections to invert** list.

5 Click **OK**.



6 Right-click **Complement 1** and choose **Rename**.

7 In the **Rename Complement** dialog box, type `Analysis domain` in the **New label** text field.

8 Click **OK**.

Infinite Element Domain 1 (ie1)

1 On the **Definitions** toolbar, click **Infinite Element Domain**.

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

3 From the **Selection** list, choose **Infinite Element domains**.

4 Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

View 1

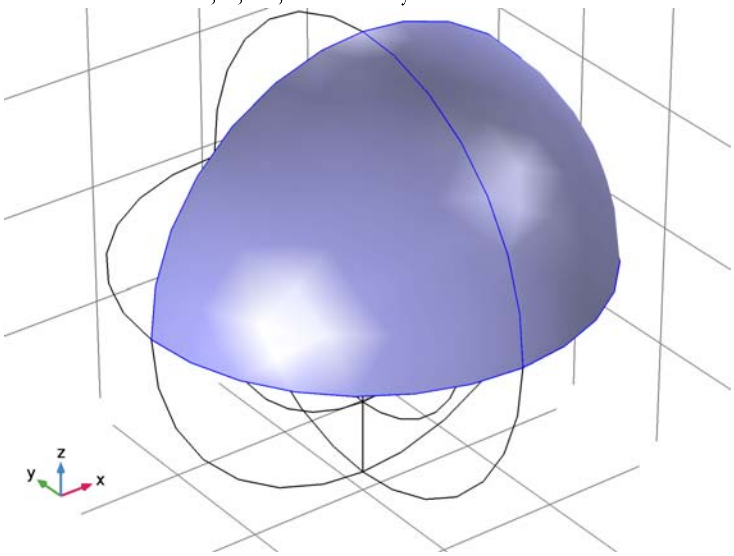
Suppress some domains to get a better view when setting up the physics and reviewing the meshed results.

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **View 1**.

Hide for Physics 1

1 On the **View 1** toolbar, click **Hide for Physics**.

2 Select Domains 2, 6, 11, and 13 only.



MAGNETIC FIELDS, NO CURRENTS (MFNC)

Set up the first physics—**Magnetic Fields, No Currents**. Begin by specifying the background magnetic field.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields, No Currents (mfnc)**.

2 In the **Settings** window for Magnetic Fields, No Currents, locate the **Background Magnetic Field** section.

3 From the **Solve for** list, choose **Reduced field**.

4 Specify the $\mathbf{H}_b$ vector as

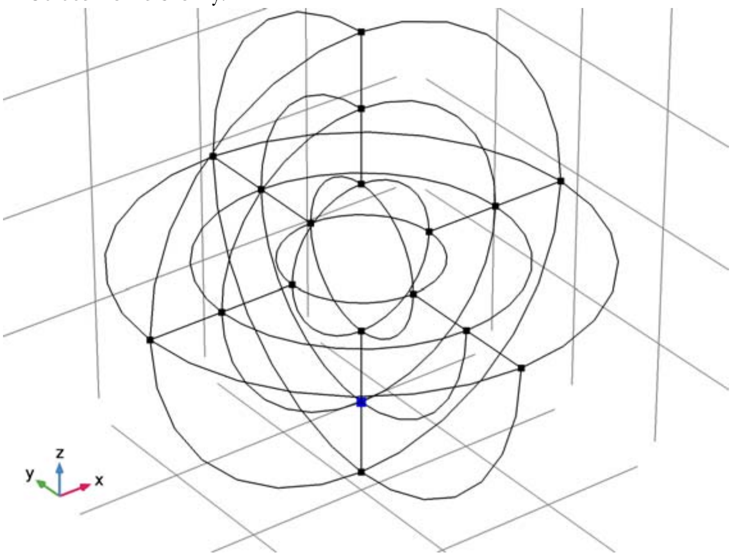
B0/mu0_const	x
0	y
0	z

Add a constraint point for the magnetic scalar potential.

Zero Magnetic Scalar Potential 1

1 On the **Physics** toolbar, click **Points** and choose **Zero Magnetic Scalar Potential**.

2 Select Point 8 only.



MATERIALS

Then, assign material properties. First, use air for all domains.

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

MATERIALS

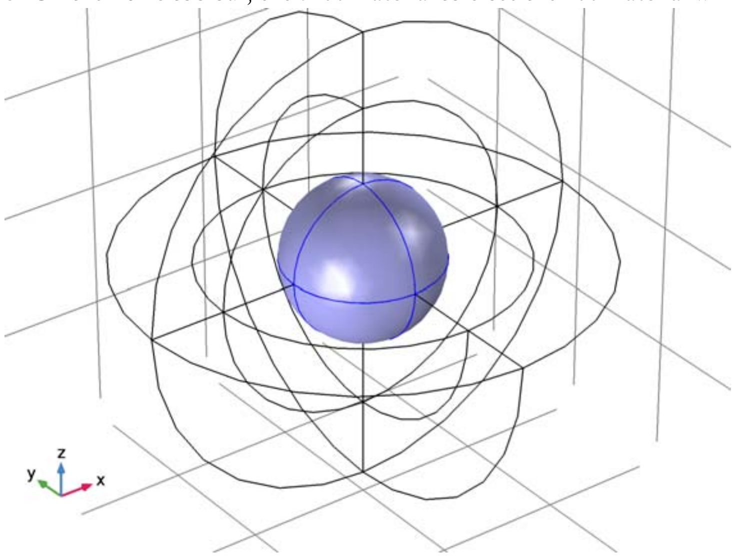
Override the core sphere with a permeable material.

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 Select Domain 9 only.

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



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

5 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permeability	mur	mu_r	1	Basic

MESH 1

Choose the finer mesh for the Analysis domain and use the swept mesh for the Infinite Element Domain.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

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

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

4 From the **Selection** list, choose **Analysis domain**.

Size

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

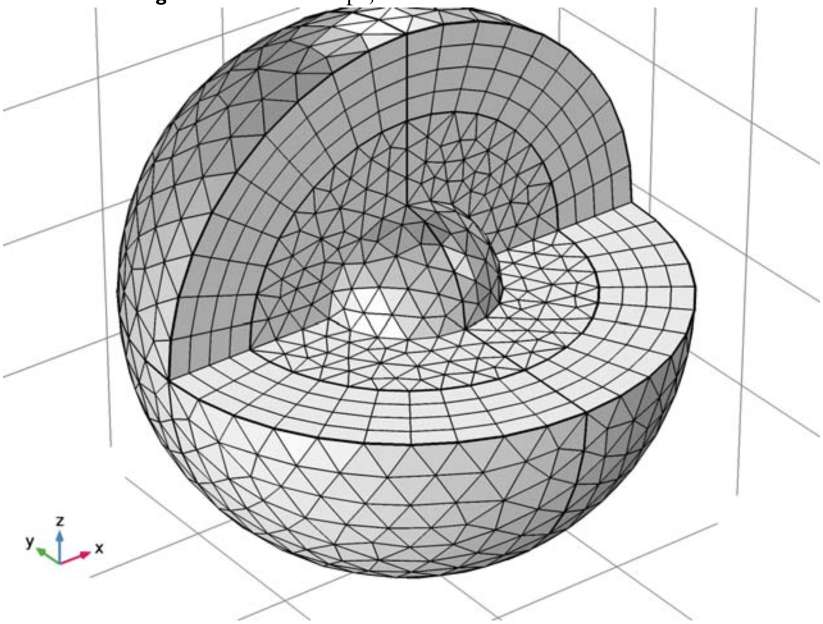
2 In the **Settings** window for Size, locate the **Element Size** section.

3 From the **Predefined** list, choose **Finer**.

Swept 1

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

2 In the **Settings** window for Swept, click **Build All**.



STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

2 In the **Settings** window for Stationary, click to expand the **Study extensions** section.

3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
mu_r	2 4 10 20 40 100 200 400 1000	

6 On the **Home** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density Norm (mfnc)

The default plot shows the magnetic flux density for all domains. By adding a selection on the current solution, you can visualize only the domain you are interested in.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/Solution 1 (sol1)**.

Selection

1 On the **Results** toolbar, click **Selection**.
2 In the **Settings** window for Selection, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **Analysis domain**.

Magnetic Flux Density Norm (mfnc)

Add an arrow plot showing the direction of the magnetic flux density.

Arrow Volume 1

1 In the **Model Builder** window, under **Results** right-click **Magnetic Flux Density Norm (mfnc)** 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 20.
4 Find the **y grid points** subsection. In the **Points** text field, type 20.
5 Find the **z grid points** subsection. In the **Points** text field, type 1.
6 On the **Magnetic Flux Density Norm (mfnc)** toolbar, click **Plot**.

   Compare the plot with Figure 2.

   Next, add a new physics, **Magnetic Fields**.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2 Go to the **Add Physics** window.
3 In the tree, select **AC/DC>Magnetic Fields (mf)**.
4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.
5 Click **Add to Component** in the window toolbar.

6 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

Set up the physics - **Magnetic Fields**. Specify the background magnetic vector potential.

MAGNETIC FIELDS (MF)

On the **Physics** toolbar, click **Magnetic Fields, No Currents (mfnc)** and choose **Magnetic Fields (mf)**.

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

2 In the **Settings** window for Magnetic Fields, locate the **Background Field** section.

3 From the **Solve for** list, choose **Reduced field**.

4 Specify the $\mathbf{A}_b$ vector as

0	x
0	y
B0*y	z

MATERIALS

Material 2 (mat2)

Assign the material properties on the core sphere.

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material 2 (mat2)**.

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

3 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Electrical conductivity	sigma	0	S/m	Basic
Relative permittivity	epsilonr	1	1	Basic

COMPONENT 1 (COMP1)

Add a mesh for the second physics.

Mesh 2

On the **Mesh** toolbar, click **Add Mesh**.

MESH 2

In the **Model Builder** window, under **Component 1 (comp1)>Meshes** right-click **Mesh 2** and choose **Free Tetrahedral**.

Free Tetrahedral 1

1 In the **Settings** window for Free Tetrahedral, locate the **Domain Selection** section.

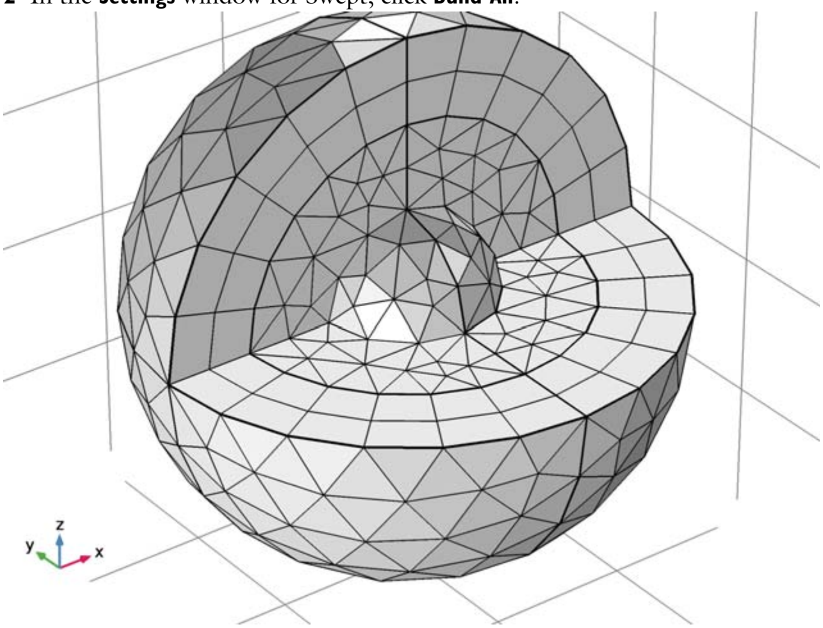
2 From the **Geometric entity level** list, choose **Domain**.

3 From the **Selection** list, choose **Analysis domain**.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 2** and choose **Swept**.

2 In the **Settings** window for Swept, click **Build All**.



STUDY 2

Step 1: Stationary

1 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check box for the **Magnetic Fields, No Currents (mfnc)** interface.

3 Click to expand the **Mesh selection** section. Locate the **Mesh Selection** section. In the table, enter the following settings:

Geometry	Mesh
Geometry 1	mesh2

4 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
mu_r	2 4 10 20 40 100 200 400 1000	

7 On the **Home** toolbar, click **Compute**.

RESULTS

Study 2/Solution 2 (sol2)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 2/Solution 2 (sol2)**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 From the **Selection** list, choose **Analysis domain**.

Data Sets

Add a Cut Point 3D data set for both physics. At the origin, the numerical and analytical results are evaluated.

Cut Point 3D 1

1 On the **Results** toolbar, click **Cut Point 3D**.

2 In the **Settings** window for Cut Point 3D, locate the **Point Data** section.

3 In the **x** text field, type `0`.
4 In the **y** text field, type `0`.
5 In the **z** text field, type `0`.

Cut Point 3D 2

1 On the **Results** toolbar, click **Cut Point 3D**.
2 In the **Settings** window for Cut Point 3D, locate the **Data** section.
3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.
4 Locate the **Point Data** section. In the **x** text field, type `0`.
5 In the **y** text field, type `0`.
6 In the **z** text field, type `0`.

Magnetic Flux Density Norm (mf)

In the **Model Builder** window, under **Results** right-click **Magnetic Flux Density Norm (mf)** and choose **Arrow Volume**.

Arrow Volume 1

1 In the **Settings** window for Arrow Volume, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields>Magnetic>mf.Bx,mf.By,mf.Bz - Magnetic flux density**.
2 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `20`.
3 Find the **y grid points** subsection. In the **Points** text field, type `20`.
4 Find the **z grid points** subsection. In the **Points** text field, type `1`.
5 On the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

   The plot should look like Figure 3.

1D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for 1D Plot Group, click to expand the **Axis** section.
3 Select the **x-axis log scale** check box.

Point Graph 1

1 On the **1D Plot Group 3** toolbar, click **Point Graph**.
2 In the **Settings** window for Point Graph, locate the **Data** section.
3 From the **Data set** list, choose **Cut Point 3D 1**.
4 Locate the **y-Axis Data** section. In the **Expression** text field, type `mfnc.normB`.

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 From the **Color** list, choose **Green**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

8 From the **Positioning** list, choose **In data points**.

1D Plot Group 3

In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

Point Graph 2

1 On the **1D Plot Group 3** toolbar, click **Point Graph**.

2 In the **Settings** window for Point Graph, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 3D 1**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `B_analytic`.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

6 From the **Color** list, choose **Black**.

1D Plot Group 3

In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

Point Graph 3

1 On the **1D Plot Group 3** toolbar, click **Point Graph**.

2 In the **Settings** window for Point Graph, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 3D 2**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `mf.normB`.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 From the **Color** list, choose **Red**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Diamond**.

8 From the **Positioning** list, choose **In data points**.

9 On the **1D Plot Group 3** toolbar, click **Plot**.

1D Plot Group 4

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

2 In the **Settings** window for 1D Plot Group, locate the **Axis** section.

3 Select the **x-axis log scale** check box.

Point Graph 1

1 On the **1D Plot Group 4** toolbar, click **Point Graph**.

2 In the **Settings** window for Point Graph, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 3D 1**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `(mfnc.normB-B_analytic)/B_analytic`.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 From the **Color** list, choose **Green**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

8 From the **Positioning** list, choose **In data points**.

1D Plot Group 4

In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

Point Graph 2

1 On the **1D Plot Group 4** toolbar, click **Point Graph**.

2 In the **Settings** window for Point Graph, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 3D 2**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `(mf.normB-B_analytic)/B_analytic`.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 From the **Color** list, choose **Red**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Diamond**.

8 From the **Positioning** list, choose **In data points**.

9 On the **1D Plot Group 4** toolbar, click **Plot**.

Compare the resulting plots with Figure 4 and Figure 5.

Created in COMSOL Multiphysics 5.2a

Quadrupole Mass Filter

Introduction

A quadrupole mass filter (QMF) is a key component of a modern mass spectrometer. A QMF uses direct current (DC) and alternating current (AC) electric fields to analyze positive or negative ions by mass-to-charge ratio. A QMF consists of four parallel rods spaced equidistantly. The ratio of the rod radius to the radius of the inscribed circle is 1.148 (Ref. 1). Opposite pairs of rods are electrically connected. Adjacent rods have opposite DC potentials and their AC potentials are out of phase. Typical rod diameters are between 5 and 12 mm with rod lengths between 100 and 200 mm. The frequency of the AC component of the electric field is typically in the range 1 to 10 MHz.

Ions are created using a number of different techniques depending on the application. They are injected along the axis of the QMF. Typical ion energies at the QMF entrance aperture are 3 to 5 electron volts. The ions experience forces due to the AC and DC fields near the quadrupole axis. For certain AC and DC field values only an ion of a specific mass to charge ratio is transmitted. Ions are detected by measuring the ion current at the exit of the quadrupole.

Note: This application requires the Particle Tracing Module.

Model Definition

The cross section of a typical QMF is shown in Figure 1. The ions are injected in such a way that their initial velocity is directed only in the out-of-plane direction. As long as the ions remain in the QMF they experience forces in the x and y direction due to the DC and AC fields. The rod radius is 2.78 mm and the radius of the inscribed circle is about 2.42 mm. Since this is a 2D model, there is no rod length. However, since the ion velocity is assumed constant in the out-of-plane direction, the effective rod length can be computed by multiplying the out-of-plane ion velocity by the total simulation time. The total simulation time is 40 µs, which, for an RF frequency of 4 MHz, corresponds to 160 RF cycles. For an initial ion energy of 3 eV and an ion mass of 40 amu, this corresponds to an effective rod length of 152 mm.

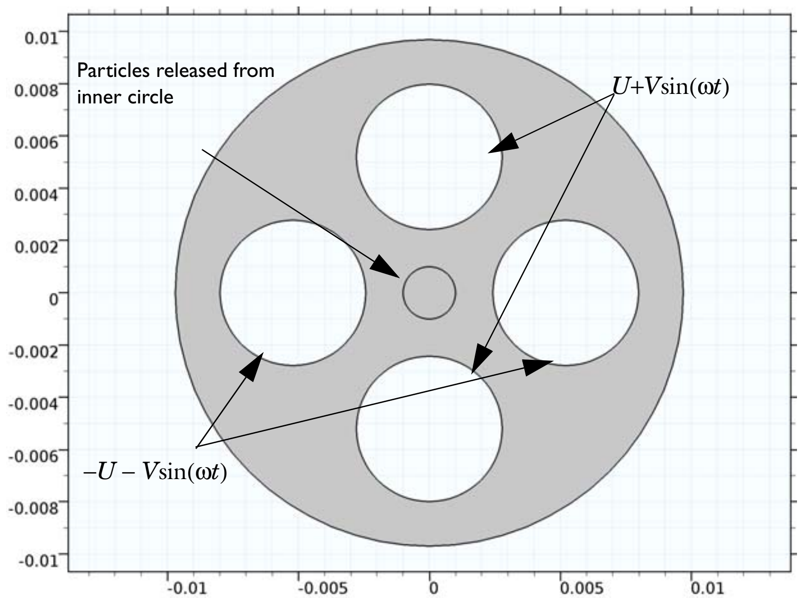


Figure 1: Plot of the model geometry. The geometry length unit is meters.

This application is designed to find the transmission probability of ions in the QMF for different values of two parameters, a and q. These are coefficients in the Mathieu equation, which can be used to solve the same problem as this application. However, this application can be generalized to 3D and the effects of fringing fields may also be included. The parameters a and q are scaled values for the DC and AC voltages respectively. It is important to note that in order to gain an accurate statistical measure of the transmission probability, it may be necessary to solve for a much larger number of particles (Ref. 1). The parameter, a, is related to the applied DC voltage by:

$$a = \frac{8eZU_{\mathrm{dc}}}{mr_0^2\omega^2}$$

where Z is the dimensionless charge number, e is the elementary charge (SI unit: s A), U_{dc} is the applied DC voltage, m is the particle mass (SI unit: kg), r_0 is the inscribed radius (SI unit: m), and ω is the angular frequency. The parameter q is related to the applied AC voltage by:

$$q = \frac{4eZV_{\text{ac}}}{mr_0^2\omega^2}$$

By solving the Mathieu equation it is possible to construct a stability diagram that shows whether the particles undergo stable or unstable oscillatory motion down the QMF. The stability diagram is plotted in Figure 2.

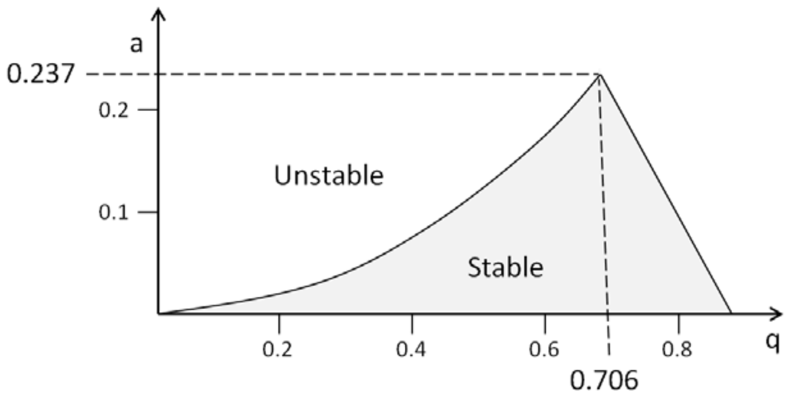


Figure 2: Stability diagram for a quadrupole mass filter.

So long as the values of a and q (and thus U_{dc} and V_{ac}) remain within the gray region, the particles do not make contact with the rods. The operating principle of a QMF is to sweep through a range of values for q whilst keeping the ratio of a/q constant. The idea is illustrated in Figure 3. The values of a and q are both increased simultaneously such that they follow the red line. The ion trajectories are initially unstable, which results in a transmission probability of zero. Once the value of q reaches about 0.6, the ions enter the stable operating region and some particles are transmitted to the detector. As the value of q increases further, the ions end up back in the unstable region and the transmission probability is reduced to zero.

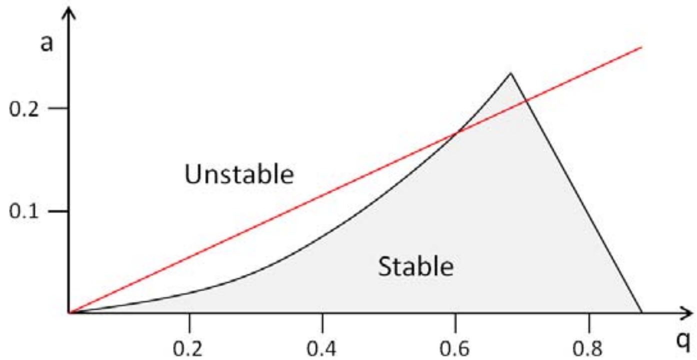


Figure 3: Operating principle of a quadrupole mass filter.

For the DC field, the application solves Poisson's equation for the electric potential, U:

$$-\nabla \cdot \varepsilon_0 \varepsilon_r \nabla U = 0 \tag{1}$$

Here ε_0 is the permittivity of free space (SI unit: F/m) and ε_r is the relative permittivity of the medium (taken as 1 in this application). The zero on the right-hand side of Equation 1 indicates that the space charge density inside the quadrupole is negligible. On the north and south rods, a positive potential of magnitude U_{dc} is applied:

$$U = U_{dc}$$

and on the east and west rods, a negative potential is applied:

$$U = -U_{dc}$$

For the AC fields, the conservation of electric currents is used to compute the AC potential, V:

$$-\nabla \cdot (\sigma + j\omega\varepsilon_0\varepsilon_r)\nabla V = 0$$

where σ is the electrical conductivity in the filter (taken as zero here) and ω is the angular frequency (SI unit: Hz).

On the north and south rods, a positive potential of magnitude V_{ac} is applied:

$$V = V_{\mathrm{ac}}$$

and on the east and west rods, a negative potential is applied:

$$V = -V_{\mathrm{ac}}$$

In order to construct the total electric field which the particles experience once they enter the modeling domain, superposition of the AC and DC fields is used. This is a valid assumption in this case since the equations solved for the AC and DC fields are linear.

Particle motion is governed by Newton's second law:

$$\frac{d}{dt}(m\mathbf{v}) = Ze\mathbf{E}$$

where m is the particle mass (SI unit: kg), $\mathbf{v}$ is the particle velocity (SI unit: m/s), Z is the dimensionless charge number, e is the elementary charge (SI unit: s A), and $\mathbf{E}$ is the electric field (SI unit: V/m). The electric field contains two contributions, a stationary electric field and one which is changing over time:

$$\mathbf{E} = \mathbf{E}_{\mathrm{dc}} + \mathbf{E}_{\mathrm{ac}}$$

where:

$$\mathbf{E}_{\mathrm{dc}} = -\nabla U$$

and

$$\mathbf{E}_{\mathrm{ac}} = -\mathrm{real}(\nabla\tilde{V}\exp(j\omega t))$$

where the tilde denotes that the AC electric potential is complex valued. The particle position, $\mathbf{q}$, is simply computed from the definition of the velocity:

$$\frac{d\mathbf{q}}{dt} = \mathbf{v}$$

The particles have no initial velocity in the modeling plane. Particles are not only released at the simulation start time, they must be released at uniformly spaced times over the first RF cycle of the AC field. Particles are released at 11 times from 0 s to 0.25 µs; since the frequency of the AC field is 4MHz, this corresponds to one full RF cycle.

The location of the ions for different values of the parameter q after 140 RF cycles are plotted in Figure 4. As expected, all the injected ions make contact with the rods up to a value of $q = 0.6$. For $q = 0.7$, however, there are ions that have clearly not made contact with the rods. These ions are detected by a current collector at the end of the QMF.

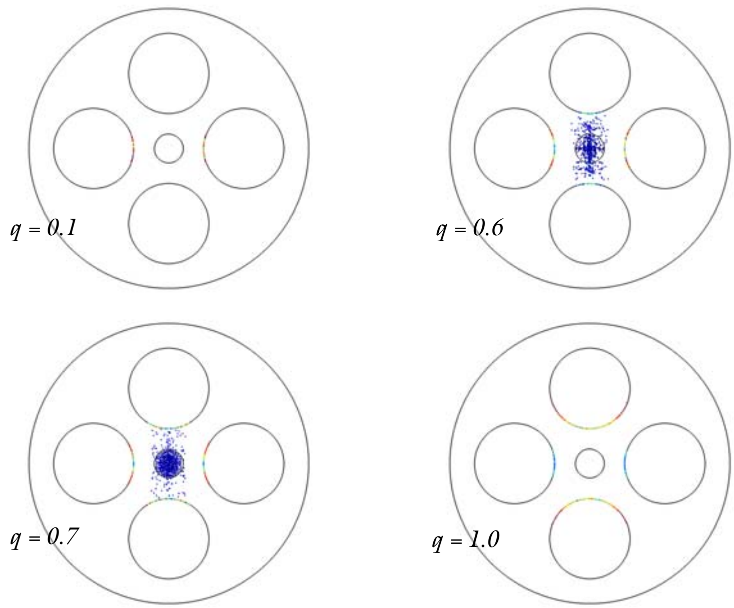


Figure 4: Plot of the ion location after 140 RF cycles inside the QMF.

The transmission probability is plotted in Figure 5. There is a clear stable window of operation between the values $q = 0.6$ and $q = 0.75$. This agrees well with the theory outlined in the Model Definition section. This example fixed the slope of a/q to be 0.2/0.7, meaning the width of the transmission peak spans over a q range of 0.15. If the slope of a/q were to be increased, the width of the peak where stable operation occurs would be reduced. If the slope of the curve is greater than 0.237/0.706 then there are no regions of stable operation for any value of q.

As the slope of a/q decreases, the range of q values for which the ion trajectories are stable increases.

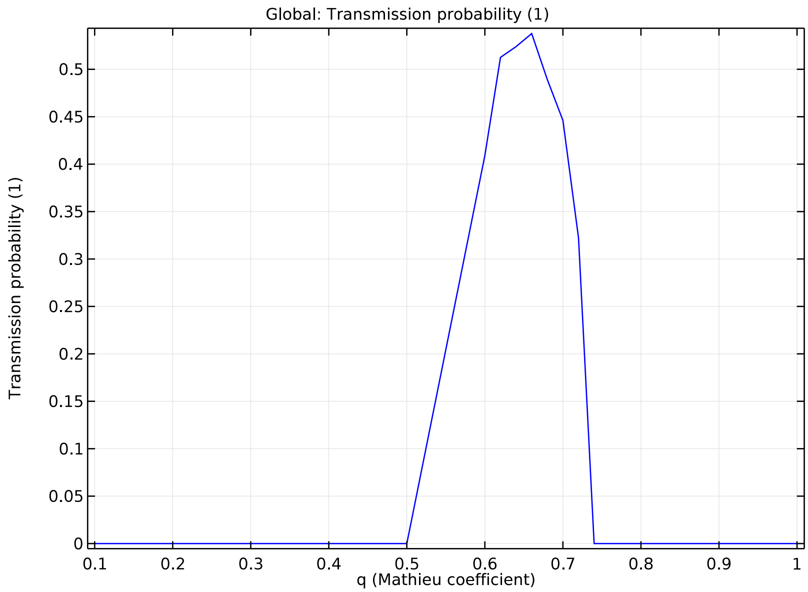


Figure 5: Plot of the transmission probability through the QMF for different values of q.

Notes About the COMSOL Implementation

Use the Parametric Sweep feature on the parameter q while keeping the ratio a/q constant. The parametric sweep wraps around studies that compute the fields and particle trajectories.

Reference

1. J.R. Gibson, S. Taylor, and J.H. Leck, "Detailed simulation of mass spectra for quadrupole mass spectrometer systems," *J. Vac. Sci. Technol. A*, vol. 18, no. 1, p. 237, 2000.

Application Library path: `ACDC_Module/Particle_Tracing/quadrupole_mass_filter`

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 **2D**.
2 In the **Select Physics** tree, select **AC/DC>Electrostatics (es)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **AC/DC>Electric Currents (ec)**.
5 Click **Add**.
6 In the **Select Physics** tree, select **AC/DC>Particle Tracing>Charged Particle Tracing (cpt)**.
7 Click **Add**.
8 Click **Done**.

GLOBAL DEFINITIONS

Add some parameters for the quadrupole geometry and physics settings. To save time, the parameters can be loaded from a file. The optimum inscribed radius is the rod radius divided by 1.147.

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the application's Application Libraries folder and double-click the file `quadrupole_mass_filter_parameters.txt`.

   The parameter `q` will be varied during a **Parametric Sweep** but the ratio of the DC and AC applied voltages will stay the same. The ionic mass is that of argon.

GEOMETRY 1

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
2 In the **Settings** window for Circle, locate the **Size and Shape** section.
3 In the **Radius** text field, type `rcase`.

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `re`.

4 Locate the **Position** section. In the **x** text field, type `re+r0`.

Circle 3 (c3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `re`.

4 Locate the **Position** section. In the **y** text field, type `re+r0`.

Circle 4 (c4)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `re`.

4 Locate the **Position** section. In the **x** text field, type `-(re+r0)`.

Circle 5 (c5)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `re`.

4 Locate the **Position** section. In the **y** text field, type `-(re+r0)`.

Circle 6 (c6)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `rsrc`.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **c1** only.

3 In the **Settings** window for Difference, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the objects **c4**, **c3**, **c2**, and **c5** only.

6 Click **Build All Objects**. The geometry should look like Figure 1.

MATERIALS

The ion trap is at near vacuum conditions. The relative permittivity is hence 1 and the conductivity 0.

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

DEFINITIONS

It is convenient to define some selections for the positive and negative electrodes.

Explicit 1

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

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

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

4 Select Boundaries 7–10, 14, 15, 18, and 19 only.

Explicit 2

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

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

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

4 Select Boundaries 3–6 and 21–24 only.

Set the **Equation form** to **Stationary** so that the correct equation contribution is generated. This is necessary because the **Frequency Domain** study step will be used to compute both the AC and DC fields later on.

ELECTROSTATICS (ES)

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

2 In the **Settings** window for Electrostatics, click to expand the **Equation** section.

3 From the **Equation form** list, choose **Stationary**.

Change the name of the dependent variable for the DC field to `U`.

4 Click to expand the **Dependent variables** section. Locate the **Dependent Variables** section. In the **Electric potential** text field, type `U`.

Now define the boundary conditions for the DC field.

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

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

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

4 Locate the **Electric Potential** section. In the V_0 text field, type `Udc`.

Electric Potential 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

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

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

4 Locate the **Electric Potential** section. In the V_0 text field, type `-Udc`.

ELECTRIC CURRENTS (EC)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents (ec)**.

2 In the **Settings** window for Electric Currents, click to expand the **Dependent variables** section.

3 Locate the **Dependent Variables** section. In the **Electric potential** text field, type `V`.

Now define the boundary conditions for the AC field.

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

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

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

4 Locate the **Electric Potential** section. In the V_0 text field, type `Vac`.

Electric Potential 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

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

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

4 Locate the **Electric Potential** section. In the V_0 text field, type `-Vac`.

Specify the properties of the ions. The charge number remains at 1 and the ion mass is set by a parameter which was defined eariler.

CHARGED PARTICLE TRACING (CPT)

Particle Properties 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Charged Particle Tracing (cpt)** node, then click **Particle Properties 1**.
2 In the **Settings** window for Particle Properties, locate the **Charge Number** section.
3 In the Z text field, type `1`.
4 Locate the **Particle Mass** section. In the m_p text field, type `mi`.
5 In the **Model Builder** window, click **Charged Particle Tracing (cpt)**.
6 In the **Settings** window for Charged Particle Tracing, locate the **Advanced Settings** section.
7 In the **Maximum number of secondary particles** text field, type `0`.

The particles are released based on the mesh elements in the aperture. Furthermore, the particles are released at different phases of the first RF cycle.

Release 1

1 On the **Physics** toolbar, click **Domains** and choose **Release**.
2 Select Domain 2 only.
3 In the **Settings** window for Release, locate the **Release Times** section.
4 Click **Range**.
5 In the **Range** dialog box, choose **Number of values** from the Entry method list.
6 In the **Start** text field, type `0`.
7 In the **Stop** text field, type `1/4E6`.
8 In the **Number of values** text field, type `11`.
9 Click **Replace**.

   There are two forces acting on the particles. One is due to the DC electric field and one is due to the AC electric field. To add these, simply add two **Electric Force** features. The **Use piecewise polynomial recovery** check box is selected so that the electric field is as accurate as possible.

Electric Force 1

1 On the **Physics** toolbar, click **Domains** and choose **Electric Force**.
2 Select Domains 1 and 2 only.

3 In the **Settings** window for Electric Force, locate the **Electric Force** section.

4 From the $\mathbf{E}$ list, choose **Electric field (es/ccn1)**.

5 Locate the **Advanced Settings** section. Select the **Use piecewise polynomial recovery on field** check box.

For the AC force, the **Multiply force by phase angle** check box must be selected. This ensures that the magnitude of the electric field is multiplied by $e^{j\omega t}$ as the particles are travelling down the trap.

Electric Force 2

1 On the **Physics** toolbar, click **Domains** and choose **Electric Force**.

2 Select Domains 1 and 2 only.

3 In the **Settings** window for Electric Force, locate the **Electric Force** section.

4 From the $\mathbf{E}$ list, choose **Electric field (ec/cucn1)**.

5 Locate the **Advanced Settings** section. Select the **Multiply force by phase angle** check box.

6 Select the **Use piecewise polynomial recovery on field** check box.

MESH 1

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

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

3 From the **Element size** list, choose **Extra fine**.

4 Click **Build All**.

Now add the study to compute the fields and the particle trajectories. Solve the problem using two study steps. First, compute the AC and DC electric fields. Then, using the computed fields to define the electric force, solve for the particle trajectories. These steps are wrapped with a parametric sweep over the parameter q. The number of parameters in the sweep is increased between the limits of 0.6 and 0.8 since this is where the transmission probability will be highest.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Empty Study**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
q	0.1 0.5 range(0.6,0.2/10,0.8) 1	

Frequency Domain

On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>Frequency Domain**.

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type `4E6`.

3 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Charged Particle Tracing**.

Time Dependent

On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

Step 2: Time Dependent

1 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

2 Click **Range**.

3 In the **Range** dialog box, type `160/4e6` in the **Stop** text field.

4 In the **Step** text field, type `1/4e6`.

5 Click **Replace**.

6 In the **Settings** window for Time Dependent, locate the **Physics and Variables Selection** section.

7 In the table, clear the **Solve for** check box for **Electrostatics** and **Electric Currents**.

8 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

9 From the **Method** list, choose **Solution**.

10 From the **Study** list, choose **Study 1, Frequency Domain**.

11 On the **Study** toolbar, click **Compute**.

RESULTS

Electric Potential (es)

Three plots are created by default. The first two show the electric potential distributions for the AC and DC fields.

Particle Trajectories (cpt)

1 In the **Model Builder** window, under **Results** click **Particle Trajectories (cpt)**.

2 In the **Settings** window for 2D Plot Group, locate the **Data** section.

3 From the **Parameter value (q)** list, choose **0.1**.

4 On the **Particle Trajectories (cpt)** toolbar, click **Plot**.

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

The particle positions at the final time are plotted for $q = 0.1$. To reproduce the remaining plots in Figure 4, select the values **0.6**, **0.7**, and **1.0** from the **Parameter value (q) list**, then click **Plot**.

Particle 1

In order to compute the transmission probability, it is necessary to add a second **Particle** data set, and then add a **Selection** to it. The number of particles counted on this selection divided by the total number of particles is the transmission probability for the selection. This value is stored in a variable called `alpha`. The fraction of particles that do not strike the rods, `1-alpha`, is the transmission probability for the QMF.

1 In the **Model Builder** window, expand the **Results>Data Sets** node.

2 Right-click **Particle 1** and choose **Duplicate**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

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

5 Select Boundaries 1–10, 13–15, and 18–24 only.

Now plot the transmission probability of the QMF as a function of `q`.

1D Plot Group 4

1 On the **Results** toolbar, click **1D Plot Group**.

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

3 Locate the **Data** section. From the **Data set** list, choose **Particle 2**.

4 From the **Time selection** list, choose **Last**.

Global 1

1 On the **Transmission Probability** toolbar, click **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
`1-cpt.alpha`	1	`Transmission probability`

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

5 Click to expand the **Legends** section. Clear the **Show legends** check box.

6 On the **Transmission Probability** toolbar, click **Plot**. Compare the resulting plot to Figure 5.

Created in COMSOL Multiphysics 5.2a

Quadrupole Mass Spectrometer

Introduction

The principal component of a quadrupole mass spectrometer is the mass filter which is used to filter ions with different charge-to-mass ratios. The quadrupole mass filter has been studied over many years (Ref. 1) and the physics and optimal design are well understood. In a real quadrupole mass spectrometer, fringe fields exist at both the entrance and exit of the mass filter. These fringe fields can play an important role in determining the transmission probability of a specific ion through the mass filter. This model computes the ion trajectories in a quadrupole mass spectrometer, including the effects of fringe fields.

Note: This application requires the Particle Tracing Module.

Model Definition

For the DC field, Poisson's equation is solved for the electric potential, U:

$$-\nabla \cdot \varepsilon_0 \varepsilon_r \nabla U = 0 \tag{1}$$

where ε_0 is the permittivity of free space (SI unit: F/m) and ε_r is the relative permittivity (taken as 1 in this model). The zero on the right-hand side of Equation 1 indicates that the space charge density inside the quadrupole is negligible (Ref. 1). On the north and south rods, a positive potential of magnitude U_{dc} is applied:

$$U = U_{dc}$$

and on the east and west rods, a negative potential is applied:

$$U = -U_{dc}$$

A small DC bias is applied on the ion aperture to help accelerate the ions into the mass filter:

$$U = U_{bias}$$

where U_{bias} is taken to be 3 V.

For the AC fields, the conservation of electric currents is used to compute the AC potential, V:

$$-\nabla \cdot (\sigma + j\omega\varepsilon_0\varepsilon_r)\nabla V = 0$$

where σ is the electrical conductivity in the mass filter (taken as zero here) and ω is the angular frequency (SI unit: Hz).

On the north and south rods, a positive potential of magnitude V_{ac} is applied:

$$V = V_{\mathrm{ac}}$$

and on the east and west rods, a negative potential is applied:

$$V = -V_{\mathrm{ac}}$$

A small AC bias is applied on the ion aperture to help accelerate the ions into the mass filter:

$$V = V_{\mathrm{bias}}$$

where V_{bias} is taken to be 3 V.

To construct the total electric field which the particles experience once they enter the modeling domain, the model uses superposition of the AC and DC fields. This is a valid assumption in this case because the equations solved for the AC and DC fields are linear.

Newton's second law governs the particle motion:

$$\frac{d}{dt}(m\mathbf{v}) = Ze\mathbf{E}$$

Here m is the particle mass (SI unit: kg), $\mathbf{v}$ is the particle velocity (SI unit: m/s), Z is the dimensionless charge number, e is the elementary charge (SI unit: s A), and $\mathbf{E}$ is the electric field (SI unit: V/m). The electric field contains two contributions, a stationary electric field and one which is changing over time:

$$\mathbf{E} = \mathbf{E}_{\mathrm{dc}} + \mathbf{E}_{\mathrm{ac}}$$

where

$$\mathbf{E}_{\mathrm{dc}} = -\nabla U$$

and

$$\mathbf{E}_{\mathrm{ac}} = -\mathrm{real}(\nabla\tilde{V}\exp(j\omega t))$$

where the tilde denotes that the AC electric potential is complex valued. The particle position, $\mathbf{q}$, is simply computed from the definition of the velocity:

$$\frac{d\mathbf{q}}{dt} = \mathbf{v}$$

The particles are released from the ion aperture with a thermal velocity the equivalent of 2 electron volts, Ref. 1. The velocity only has an x-component:

$$v_x = \sqrt{\frac{2eA}{m}}$$

where A is 2 eV. Particles are not only released at the simulation start time, they must be released at uniformly spaced times over the first RF cycle of the AC field. Particles are released at 11 times beginning at 0 s and ending at 0.25 µs. Because the frequency of the AC field is 4 MHz, this corresponds to one full RF cycle.

Results and Discussion

The particle trajectories are plotted in. It is obvious from the plot that the ion transmission probability is very high, 100% actually. This is because a very stable operating point on the a-q curve has been chosen. The ions remain in the quadrupole mass filter for around 140 RF cycles.

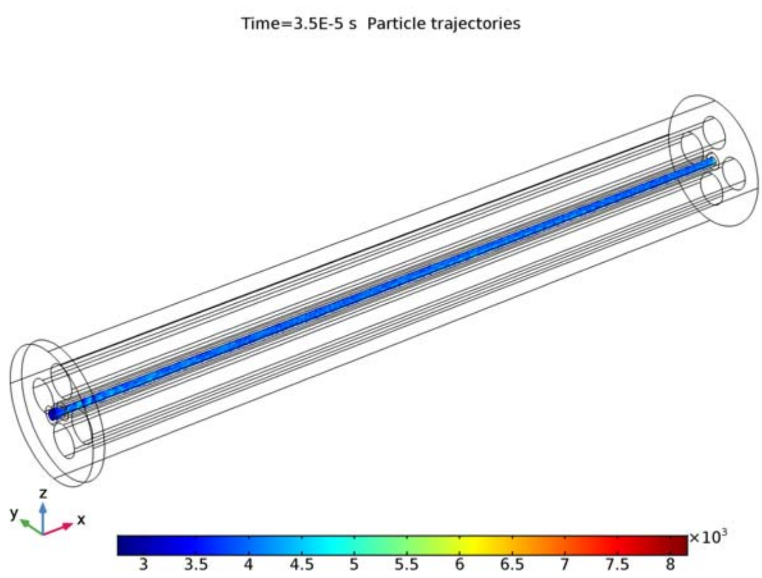


Figure 1: Plot of the particle trajectories inside the quadrupole. The color represents the particle velocity (m/s).

Due to the presence of the biased plate surrounding the ion aperture, the ions gain energy as they move through the quadrupole. This can be seen in Figure 2; the ions have a mean energy of 5 eV over a range of around 3 eV. The spread in energy can be attributed to the fact that there is a small DC and AC bias. The AC bias can be positive or negative which accelerates or decelerates the ions depending on the phase of the RF cycle when they are released.

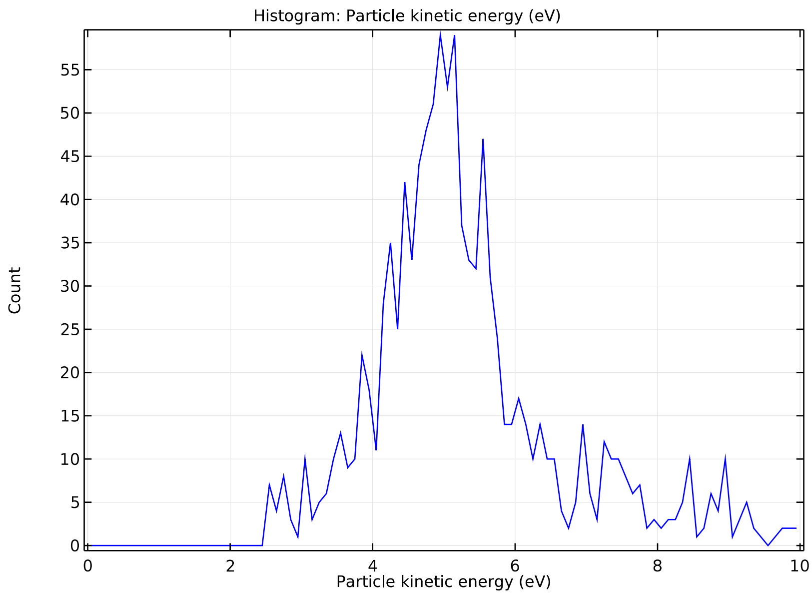


Figure 2: Plot of the energy distribution of the ions at the collector.

Notes About the COMSOL Implementation

The model is solved in two stages. First, the DC and AC fields are computed. Then the particle trajectories are computed and their motion is driven by the AC and DC electric fields.

Reference

1. J.R. Gibson, S. Taylor, and J.H. Leck, "Detailed simulation of mass spectra for quadrupole mass spectrometer systems," *J. Vac. Sci. Technol. A*, vol. 18, no. 1, p. 237, 2000.

Application Library path: `ACDC_Module/Particle_Tracing/quadrupole_mass_spectrometer`

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>Electrostatics (es)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **AC/DC>Electric Currents (ec)**.

5 Click **Add**.

6 In the **Select Physics** tree, select **AC/DC>Particle Tracing>Charged Particle Tracing (cpt)**.

7 Click **Add**.

8 Click **Done**.

GLOBAL DEFINITIONS

Add some parameters for the quadrupole geometry and physics settings. To save time, the parameters can be loaded from a file.

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 Click **Load from File**.

4 Browse to the application's Application Libraries folder and double-click the file `quadrupole_mass_spectrometer_parameters.txt`.

GEOMETRY 1

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for Work Plane, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

4 Click **Show Work Plane**.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `re`.

4 Locate the **Position** section. In the **xw** text field, type `-(re+r0)`.

Circle 2 (c2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `re`.

4 Locate the **Position** section. In the **xw** text field, type `re+r0`.

Circle 3 (c3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `re`.

4 Locate the **Position** section. In the **yw** text field, type `-(re+r0)`.

Circle 4 (c4)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `re`.

4 Locate the **Position** section. In the **yw** text field, type `re+r0`.

5 On the **Work Plane** toolbar, click **Build All**.

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

Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for Extrude, locate the **Distances from Plane** section.

3 In the table, enter the following settings:

Distances (m)
`Lquad`

4 Click **Build All Objects**.

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

Work Plane 2 (wp2)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for Work Plane, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

4 In the **x-coordinate** text field, type `-fd`.

5 Click **Show Work Plane**.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `rcase`.

Circle 2 (c2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `rsrc`.

Circle 3 (c3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `2*rsrc`.

4 On the **Work Plane** toolbar, click **Build All**.

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

Work Plane 2 (wp2)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 2 (wp2)**.

Extrude 2 (ext2)

1 On the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for Extrude, locate the **Distances from Plane** section.

3 In the table, enter the following settings:

Distances (m)
`fd`
`Lquad`

4 Click **Build All Objects**.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **ext2** only.

3 In the **Settings** window for Difference, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **ext1** only.

6 Click **Build All Objects**.

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

Now define some selections for the positively and negatively charged rods.

DEFINITIONS

Explicit 1

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

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

3 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

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

5 Select Boundaries 25, 26, 28, 29, 37, 38, 43, and 44 only.

Explicit 2

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

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

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

4 Select Boundaries 20–23 and 47–50 only.

ELECTROSTATICS (ES)

Define the boundary conditions for the electrostatics problem.

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

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

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

4 Locate the **Electric Potential** section. In the V_0 text field, type `Udc`.

Electric Potential 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

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

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

4 Locate the **Electric Potential** section. In the V_0 text field, type `-Udc`.

Electric Potential 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 Select Boundaries 1, 4, and 7 only.

3 In the **Settings** window for Electric Potential, locate the **Electric Potential** section.

4 In the V_0 text field, type `3`.

ELECTRIC CURRENTS (EC)

Now define the boundary conditions for the AC part of the problem.

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Electric Currents (ec)** node, then click **Electric Currents (ec)**.

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

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

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

4 Locate the **Electric Potential** section. In the V_0 text field, type `Vac`.

Electric Potential 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

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

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

4 Locate the **Electric Potential** section. In the V_0 text field, type `-Vac`.

Electric Potential 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 Select Boundaries 1, 4, and 7 only.

3 In the **Settings** window for Electric Potential, locate the **Electric Potential** section.

4 In the V_0 text field, type `3`.

Specify the mass and charge number of the ions.

CHARGED PARTICLE TRACING (CPT)

Particle Properties 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Charged Particle Tracing (cpt)** node, then click **Particle Properties 1**.

2 In the **Settings** window for Particle Properties, locate the **Particle Mass** section.

3 In the m_p text field, type `mi`.

4 Locate the **Charge Number** section. In the Z text field, type `1`.

The particles are released from a projected plane grid on the **Inlet** boundary. There are 100 particles per release and 11 releases in total. The particles all move in the x-direction with initial energy 2 eV.

5 In the **Model Builder** window, click **Charged Particle Tracing (cpt)**.

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 7 only.

3 In the **Settings** window for Inlet, locate the **Initial Position** section.

4 From the **Initial position** list, choose **Projected plane grid**.

5 In the N text field, type `100`.

6 Locate the **Release Times** section. Click **Range**.

7 In the **Range** dialog box, choose **Number of values** from the Entry method list.

8 In the **Start** text field, type `0`.

9 In the **Stop** text field, type `1/4E6`.

10 In the **Number of values** text field, type `11`.

11 Click **Replace**.

12 In the **Settings** window for Inlet, locate the **Initial Velocity** section.

13 Specify the $\mathbf{v}_0$ vector as

vx0	x
0	y
0	z

Add an **Electric Force** feature for the DC field.

Electric Force 1

1 On the **Physics** toolbar, click **Domains** and choose **Electric Force**.

2 In the **Settings** window for Electric Force, locate the **Electric Force** section.

3 From the $\mathbf{E}$ list, choose **Electric field (es/ccn1)**.

4 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.

Add another **Electric Force** feature for the AC field.

Electric Force 2

1 On the **Physics** toolbar, click **Domains** and choose **Electric Force**.

2 In the **Settings** window for Electric Force, locate the **Electric Force** section.

3 From the $\mathbf{E}$ list, choose **Electric field (ec/cucn1)**.

4 Locate the **Advanced Settings** section. Select the **Multiply force by phase angle** check box.

5 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

Material 1 (mat1)

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

2 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	1	1	Basic
Electrical conductivity	sigma	0	S/m	Basic

Since the aspect ratio of the quadrupole is very high, it is more efficient to use a swept mesh. This is allowed in this case because the field does not change in the x-direction once the particles have passed through the fringing fields.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

Free Triangular 1

Select Boundaries 16, 19, 24, 27, 30, 33, and 46 only.

Size 1

1. Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Size**.
2. In the **Settings** window for Size, locate the **Element Size** section.
3. From the **Predefined** list, choose **Extra fine**.
4. Click to expand the **Element size parameters** section. Locate the **Element Size** section. Click the **Custom** button.
5. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
6. Select the **Minimum element size** check box.
7. Select the **Curvature factor** check box.
8. In the associated text field, type `0.15`.
9. Click **Build All**.

Swept 1

1. In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.
2. In the **Settings** window for Swept, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 4–6 only.

Distribution 1

1. Right-click **Component 1 (comp1)>Mesh 1>Swept 1** and choose **Distribution**.
2. In the **Settings** window for Distribution, locate the **Distribution** section.
3. From the **Distribution properties** list, choose **Predefined distribution type**.
4. In the **Number of elements** text field, type `35`.
5. In the **Element ratio** text field, type `15`.
6. Click **Build All**.
7. Click the **Go to Default 3D View** button on the **Graphics** toolbar.

Swept 2

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

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

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

4 Select Domains 1–3 only.

5 Right-click **Component 1 (comp1)>Mesh 1>Swept 2** and choose **Distribution**.

Distribution 1

In the **Settings** window for Distribution, click **Build All**.

Add a **Stationary** study to compute the electrostatic field.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Custom Studies>Preset Studies for Some Physics Interfaces>Stationary**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1

Step 1: Stationary

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

2 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Electric Currents** and **Charged Particle Tracing**.

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

5 In the **Rename Study** dialog box, type `Electrostatic Study` in the **New label** text field.

6 Click **OK**.

Add a **Frequency Domain** study to compute the AC field.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Custom Studies>Preset Studies for Some Physics Interfaces>Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2** 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 `4[MHz]`.

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Electrostatics** and **Charged Particle Tracing**.

5 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

6 From the **Method** list, choose **Solution**.

7 From the **Study** list, choose **Electrostatic Study, Stationary**.

8 In the **Model Builder** window, right-click **Study 2** and choose **Rename**.

9 In the **Rename Study** dialog box, type `Electric currents Study` in the **New label** text field.

10 Click **OK**.

Add a **Time Dependent** study to compute the particle trajectories.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Time Dependent**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 3** click **Step 1: Time Dependent**.

2 In the **Settings** window for Time Dependent, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Electrostatics** and **Electric Currents**.

4. Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
5. From the **Method** list, choose **Solution**.
6. From the **Study** list, choose **Electric currents Study, Frequency Domain**.
7. Locate the **Study Settings** section. In the **Times** text field, type `range(0,1/4e6,140/4e6)`.
8. Click **Range**.
9. In the **Range** dialog box, click **Replace**.
10. In the **Model Builder** window, right-click **Study 3** and choose **Rename**.
11. In the **Rename Study** dialog box, type `Particle tracing Study` in the **New label** text field.
12. Click **OK**.

ELECTROSTATIC STUDY

Compute the DC field.

1. On the **Home** toolbar, click **Compute**.

ELECTRIC CURRENTS STUDY

Compute the AC field.

1. Click **Compute**.

PARTICLE TRACING STUDY

Now, compute the particle trajectories.

1. Click **Compute**.

RESULTS

Particle Trajectories (cpt)

1. In the **Model Builder** window, under **Results** click **Particle Trajectories (cpt)**.
2. In the **Settings** window for 3D Plot Group, click to expand the **Color legend** section.
3. Locate the **Color Legend** section. From the **Position** list, choose **Bottom**.

Particle Trajectories 1

1. In the **Model Builder** window, expand the **Particle Trajectories (cpt)** node, then click **Particle Trajectories 1**.

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

3 Find the **Line style** subsection. From the **Type** list, choose **Line**.

4 Find the **Point style** subsection. From the **Type** list, choose **None**.

5 Find the **Line style** subsection. From the **Interpolation** list, choose **Uniform**.

6 In the **Number of interpolated times** text field, type `1000`.

DEFINITIONS

View 1

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

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

3 Clear the **Show grid** check box.

RESULTS

Particle Trajectories (cpt)

1 In the **Model Builder** window, under **Results** click **Particle Trajectories (cpt)**.

2 On the **Particle Trajectories (cpt)** toolbar, click **Plot**.

3 Click the **Zoom Extents** button on the **Graphics** toolbar. The resulting plot should look like Figure 1.

Finally, create a **Histogram** of the ion energy distribution function at the time all the ions have reached the current collector.

1D Plot Group 4

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

2 In the **Settings** window for 1D Plot Group, type `Ion Energy Distribution Function` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Particle 1**.

4 From the **Time selection** list, choose **Last**.

Histogram 1

1 On the **Ion Energy Distribution Function** toolbar, click **Histogram**.

2 In the **Settings** window for Histogram, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Charged Particle Tracing>Velocity and energy>cpt.Ep - Particle kinetic energy**.

3 Locate the **Expression** section. From the **Unit** list, choose **eV**.

4 Locate the **Bins** section. From the **Entry method** list, choose **Limits**.

5 Click **Range**.

6 In the **Range** dialog box, choose **Number of values** from the Entry method list.

7 In the **Start** text field, type `0`.

8 In the **Stop** text field, type `10`.

9 In the **Number of values** text field, type `101`.

10 Click **Replace**.

11 On the **Ion Energy Distribution Function** toolbar, click **Plot**. The resulting plot should look like Figure 2.

Created in COMSOL Multiphysics 5.2a

An RFID System

Introduction

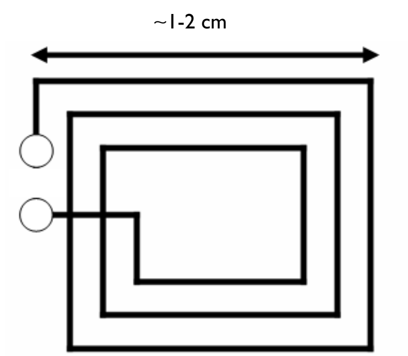
This example illustrates the modeling of a reader-transponder pair for radio-frequency identification (RFID) applications.

An RFID system consists of two main parts:

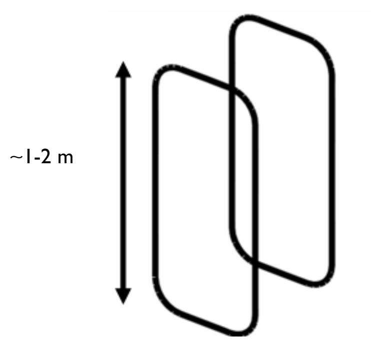
- A tag or transponder with a printed circuit-board (PCB) antenna
- A reader unit with a larger RF antenna

The reader antenna generates an electromagnetic field that energizes a chip (IC-circuit) inside the tag. The electromagnetic field is modulated by the tag's circuit and the modulated signal is recovered by the reader antenna.

The transponder antenna is typically a conductor pattern on a substrate:



and a common type of reader antenna is a larger dual coil:



INDUCTIVE COUPLING

The coupling of the antennas is mainly inductive and is characterized by the mutual inductance, denoted L_{12}. The mutual inductance is defined as the total magnetic flux intercepted by one antenna for a unit current flowing in the other antenna

$$L_{12} = \frac{\iint_{S_2} \mathbf{B} \cdot \mathbf{n} dS}{I_1} \tag{1}$$

where S_2 is the area of coil number 2, $\mathbf{B}$ is the flux intercepted by coil 2, I_1 is the current running in coil number 1, $\mathbf{n} = (n_x, n_y, n_z)$ is the unit surface normal vector, and dS is an infinitesimal area element.

It is possible to transform the surface integral in Equation 1 into a simpler line integral by using the magnetic vector potential

$$\mathbf{B} = \nabla \times \mathbf{A}$$

together with Stokes' theorem, which states that a surface integral of the curl of a field equals the closed line integral over the rim of the surface:

$$L_{12} = \frac{\iint_{S_2} (\nabla \times \mathbf{A}) \cdot \mathbf{n} dS}{I_1} = \frac{\oint_{\Gamma_S} \mathbf{A} \cdot \mathbf{t} dl}{I_1}$$

Here $\mathbf{t} = (t_x, t_y, t_z)$ is the unit tangent vector of the curve Γ_S and dl is an infinitesimal line element.

Because the coupling is dominated by near-field inductive effects, it is sufficient to compute the mutual inductance for the static case (frequency equals zero) and neglect capacitive effects along with wave propagation phenomena. The appropriate physics to use is called Magnetic Fields and is available in the AC/DC Module. It has the magnetic vector potential $\mathbf{A} = (A_x, A_y, A_z)$ as its unknown field quantity.

EDGE COMPUTATIONS

This example approximates coils and wires as edges (1D entities) embedded in 3D space and the variables and techniques used are somewhat different than for a regular magnetostatics boundary value problem. The magnetic vector potential components are available at edges as the variables `tAx`, `tAy`, and `tAz`, which form the projection of the vector potential on edges. On the other hand, the bulk components are denoted `Ax`, `Ay`, and `Az`.

Results

The computed value for the mutual inductance L_{12} is 0.99 nH. Figure 1shows magnetic flux lines and the base-10 logarithm of the magnetic flux intensity as colors.

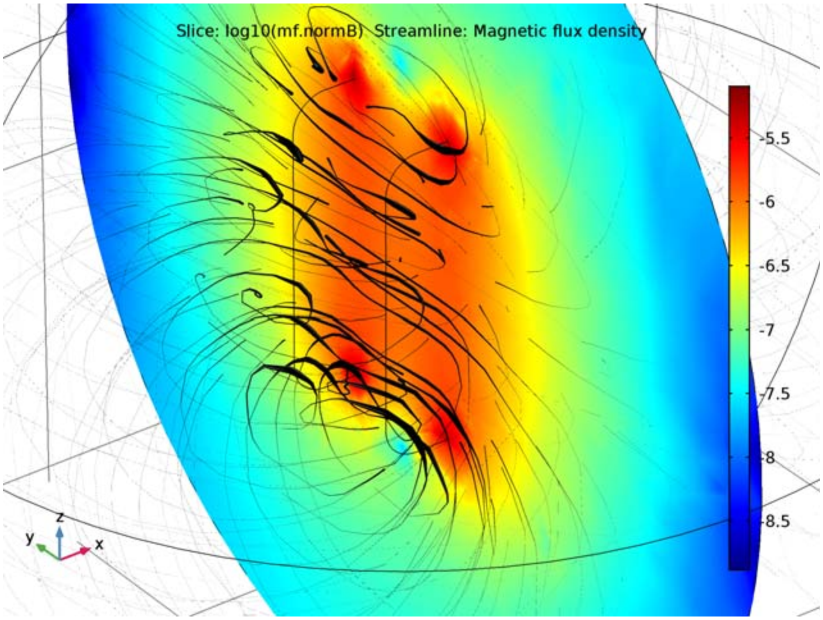


Figure 1: Magnetic flux density (logarithmic scale) and flux lines. The contours of the reader antennas are visible in the center of the plot.

Application Library path: `ACDC_Module/Inductive_Devices_and_Coils/rfid`

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>Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

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

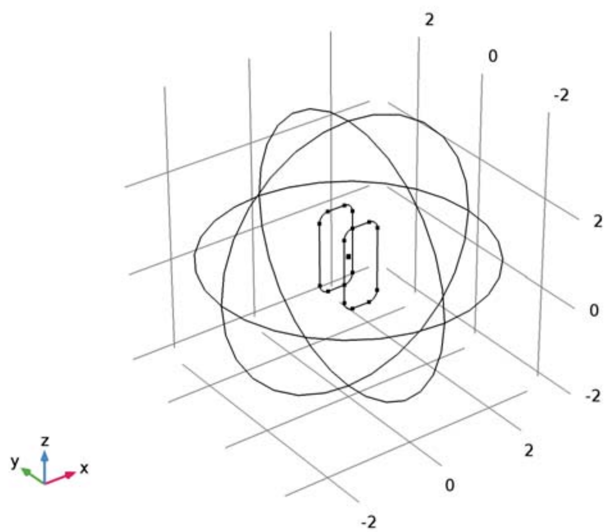
1 On the **Geometry** toolbar, click **Insert Sequence**.

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

3 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Build All Objects**.

4 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

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



DEFINITIONS

This section asks you to define a number of edge selections for subsequent use in setting up the physics and postprocessing.

Explicit 1

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

2 In the **Model Builder** window, right-click **Explicit 1** and choose **Rename**.
3 In the **Rename Explicit** dialog box, type `Transponder` in the **New label** text field.
4 Click **OK**.
5 In the **Settings** window for Explicit, locate the **Input Entities** section.
6 From the **Geometric entity level** list, choose **Edge**.
7 Zoom in a few times so that you can clearly see the transponder antenna.
8 Click the **Select Box** button on the **Graphics** toolbar.
9 Use the Select Box tool to select the transponder antenna. Verify that the selected edges are 15-23, 29-34, and 38-40.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Model Builder** window, right-click **Explicit 2** and choose **Rename**.
3 In the **Rename Explicit** dialog box, type `Reader 1` in the **New label** text field.
4 Click **OK**.
5 In the **Settings** window for Explicit, locate the **Input Entities** section.
6 From the **Geometric entity level** list, choose **Edge**.
7 Click the **Zoom Out** button on the **Graphics** toolbar.
8 Click the **Zoom Out** button on the **Graphics** toolbar.
9 Select all the edges of the reader coil *behind* the transponder. Verify that you have selected edges 8-10, 13, 14, 43, 44, and 46 only.

Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Model Builder** window, right-click **Explicit 3** and choose **Rename**.
3 In the **Rename Explicit** dialog box, type `Reader 2` in the **New label** text field.
4 Click **OK**.
5 In the **Settings** window for Explicit, locate the **Input Entities** section.
6 From the **Geometric entity level** list, choose **Edge**.
7 Select all the edges of the reader *in front of* the transponder. Verify that you have selected edges 5-7, 11, 12, 41, 42, and 45.

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MAGNETIC FIELDS (MF)

Edge Current 1

1 On the **Physics** toolbar, click **Edges** and choose **Edge Current**.

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

3 From the **Selection** list, choose **Reader 1**.

4 Locate the **Edge Current** section. In the I_0 text field, type `1`.

Edge Current 2

1 On the **Physics** toolbar, click **Edges** and choose **Edge Current**.

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

3 From the **Selection** list, choose **Reader 2**.

4 Locate the **Edge Current** section. In the I_0 text field, type `-1`.

The reference direction for the **Edge Current** feature is indicated by arrows on the edge selection in the **Graphics** window. A negative value means that the edge current will flow in the direction opposite to the arrow.

The default boundary condition for the faces that constitute the sphere is **Magnetic Insulation**. This allows for surface currents to run across the sphere in such a way as to make the tangential component of the magnetic vector potential equal to zero. This is a good approximation in place of an infinite domain. It approximates that the tangential component of the vector potential approaches zero as the distance from the coils approaches infinity. Thus there is no need to change the boundary settings.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size

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

2 In the **Settings** window for Size, locate the **Element Size** section.

3 From the **Predefined** list, choose **Coarser**.

4. Click to expand the **Element size parameters** section. Locate the **Element Size Parameters** section. In the **Minimum element size** text field, type `0.0025`.
5. Click **Build All**.
6. Click the **Zoom Extents** button on the **Graphics** toolbar.

STUDY 1

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

RESULTS

Magnetic Flux Density Norm (mf)

The default plot shows the magnetic flux density norm in three cross sections. Follow the instructions to get a more informative plot of the fields in the air domain surrounding the coils.

Multislice 1

1. In the **Model Builder** window, expand the **Magnetic Flux Density Norm (mf)** node.
2. Right-click **Multislice 1** and choose **Delete**. Click **Yes** to confirm.

Magnetic Flux Density Norm (mf)

In the **Model Builder** window, under **Results** right-click **Magnetic Flux Density Norm (mf)** and choose **Slice**.

Slice 1

1. In the **Settings** window for Slice, locate the **Plane Data** section.
2. In the **Planes** text field, type `1`.
3. On the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

   The plot is dominated by the singular flux density where the reader antennas intersect the slice. You can avoid this problem by adjusting the range, but a convenient way to cover several orders of magnitude is to plot the 10-logarithm of the flux density.
4. Locate the **Expression** section. In the **Expression** text field, type `log10(mf.normB)`.
5. On the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

   Next, add flux lines to reproduce Figure 1.

Magnetic Flux Density Norm (mf)

Right-click **Magnetic Flux Density Norm (mf)** and choose **Streamline**.

Streamline 1

1. In the **Settings** window for Streamline, locate the **Streamline Positioning** section.

2 From the **Positioning** list, choose **Uniform density**.
3 In the **Separating distance** text field, type `0.05`.
4 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.
5 In the **Tube radius expression** text field, type `mf.normB`.
6 From the **Color** list, choose **Black**.
7 Click the **Zoom In** button on the **Graphics** toolbar.
8 Click the **Zoom In** button on the **Graphics** toolbar.
9 On the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

The starting points for the magnetic field lines (streamlines) are randomized, which makes them unevenly distributed. However, you can see that the field lines are qualitatively following the expected pattern.

As a final step, evaluate the mutual inductance:

Line Integration 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Line Integration**.
2 In the **Settings** window for Line Integration, locate the **Selection** section.
3 From the **Selection** list, choose **Transponder**.
4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`(tAx*t1x+tAy*t1y+tAz*t1z)/1[A]`	`nH`	`Mutual inductance`

5 Click **Evaluate**.

The result should be close to 0.99 nH.

Created in COMSOL Multiphysics 5.2a

Rotating Machinery 3D Tutorial

Introduction

This application serves as a general introduction to the Rotating Machinery, Magnetic interface in 3D. The circular motion of a cylindrical copper rotor near a stationary permanent magnet generates induced eddy currents in the rotor. Figure 1 shows the geometry with the rotor and stator.

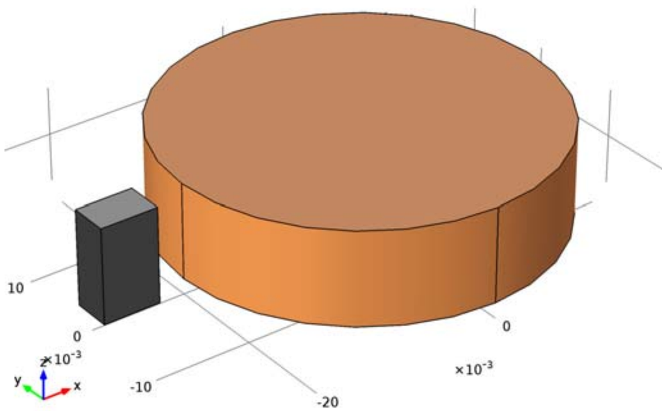


Figure 1: Drawing showing how the rotor and stator with permanent magnet are defined.

Model Definition

This COMSOL Multiphysics application is a time-dependent 3D problem. It is a true time-dependent model where the motion of the rotor is accounted for in the boundary condition between the stator and rotor geometries. The conducting part of the rotor is modeled using Ampère's law;

$$\sigma\frac{\partial\mathbf{A}}{\partial t}+\nabla\times\left(\frac{1}{\mu}\nabla\times\mathbf{A}\right) = 0$$

whereas the nonconducting parts of both the rotor and stator are modeled using a magnetic flux conservation equation for the scalar magnetic potential:

$$-\nabla\cdot(\mu\nabla V_{\mathrm{m}}-\mathbf{B}_{\mathrm{r}}) = 0$$

Rotation is modeled using a ready-made physics interface for rotating machinery. The central part of the geometry, containing the rotor and part of the air-gap, is modeled as rotating relative to the coordinate system of the stator. The rotor and the stator are created as two separate geometry objects, so it is possible to use an assembly (see the Geometry chapter in the *COMSOL Multiphysics Reference Manual* for details).

This has several advantages: the coupling between the rotor and the stator is done automatically, the parts can be meshed independently, and it allows for a controlled discontinuity in the scalar magnetic potential at the interface between the two geometry objects. The rotor problem is solved in a rotating coordinate system where the rotor is fixed (the rotor frame), whereas the stator problem is solved in a coordinate system that is fixed with respect to the stator (the stator frame). Using COMSOL notation, they are both solved in the material frame. An identity pair connecting the rotating rotor frame with the fixed stator frame is created between the rotor and the stator. The identity pair enforces continuity for the magnetic scalar potential in the global fixed coordinate system (the stator frame relative to which the rotor rotates).

However, this means that in the frame on which continuity in the scalar magnetic potential is enforced, the meshes on either side of the rotor-stator interface cannot be made identical except for the case without any rotation so some interpolation between non-conforming meshes is involved. The resulting interpolation errors have little numerical impact if the assembly is created such that the resulting identity boundary pair only involves the scalar magnetic potential. In Ampère's law for the magnetic vector potential, current conservation is an implicit requirement that is violated if the identity boundary pair would involve interpolation of the magnetic vector potential. The resulting interpolation errors unconditionally make such a model numerically unstable. Thus, special care has to be exercised when setting up the geometry using assemblies in an application like this.

Note: An additional intricacy when using a mixed potential formulation involving both scalar and vector magnetic potentials is that the domains using the scalar magnetic potential must be simply connected. A domain is simply connected if any closed line integration path does not link an external domain. An example of a not simply connected domain is a torus (as a closed loop may link the central hole). This is a requirement imposed by the integral form of Ampère's law as, for example, the hole in the torus may carry a current linking the torus. In the scalar magnetic potential formulation, closed loop line integrals of the H field must evaluate to zero.

Results and Discussion

The generated eddy current distribution in the rotor winding is shown after 0.1 s for a constant rotation speed of 600 rpm.

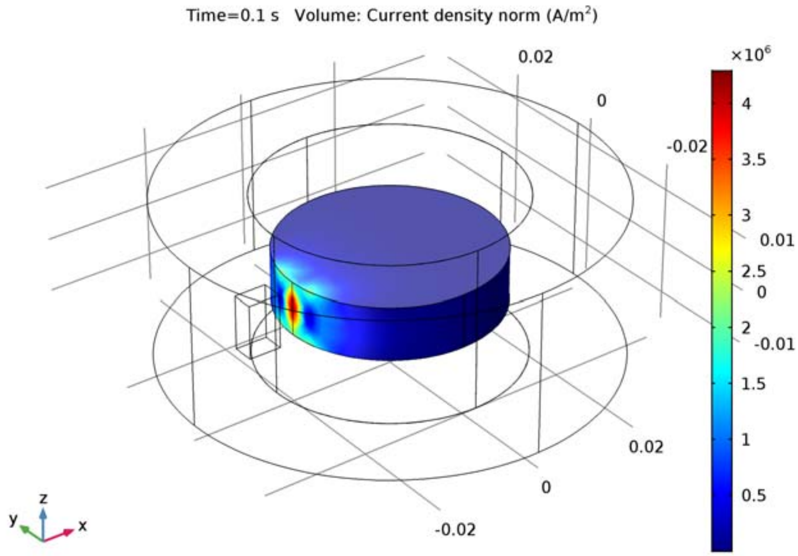


Figure 2: Eddy current distribution after 0.1 s.

Application Library path: `ACDC_Module/Motors_and_Actuators/rotating_machinery_3d_tutorial`

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>Rotating Machinery, Magnetic (rmm)**.

3 Click **Add**.

4 Click **Study**.

A stationary study is needed to compute initial conditions. You will add a time-dependent study later before solving.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`rpm`	`600[rpm]`	10 Hz	

GEOMETRY 1

The geometry must be segmented in at least two parts, the stator and the rotor, to allow relative rotation. The geometry sequence for this tutorial can be imported from the file `rotating_machinery_3d_tutorial_geom_sequence.mph`.

1 On the **Geometry** toolbar, click **Insert Sequence**.

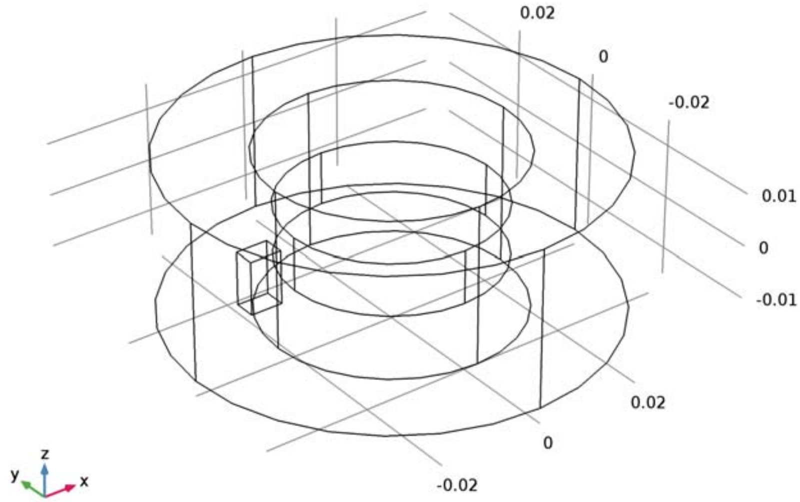
2 Browse to the application's Application Libraries folder and double-click the file `rotating_machinery_3d_tutorial_geom_sequence.mph`.

Form Assembly (fin)

1 On the **Geometry** toolbar, click **Build All**.

2 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

3 Click the **Wireframe Rendering** button on the **Graphics** toolbar.



A boundary pair is automatically created between rotor and stator.

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

ADD MATERIAL

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

2 In the tree, select **AC/DC>Copper**.

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

MATERIALS

Copper (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat2)**.

2 Select Domain 4 only.

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

Use **Magnetic Flux Conservation** in the non-conducting domains and **Ampère's Law** in the conducting domains. Set up the permanent magnet as a user defined domain with remanence and permeability.

ROTATING MACHINERY, MAGNETIC (RMM)

Magnetic Flux Conservation 1

1. On the **Physics** toolbar, click **Domains** and choose **Magnetic Flux Conservation**.
2. Select Domains 1 and 3 only.

Magnetic Flux Conservation 2

1. On the **Physics** toolbar, click **Domains** and choose **Magnetic Flux Conservation**.
2. Select Domain 2 only.
3. In the **Settings** window for Magnetic Flux Conservation, locate the **Magnetic Field** section.
4. From the μ_r list, choose **User defined**. From the **Constitutive relation** list, choose **Remanent flux density**.
5. In the μ_r text field, type `1.05`.
6. Specify the $\mathbf{B}_\mathbf{r}$ vector as

1.2	X
0	Y
0	Z

Rotating machinery in 3D needs explicit gauge fixing of the vector potential.

Gauge Fixing for A-Field 1

1. On the **Physics** toolbar, click **Domains** and choose **Gauge Fixing for A-Field**.
2. Select Domain 4 only.

The gauge fixing needs to be constrained in at least one point. You need to set this up manually because the gauge fixed domain and its boundaries are interior to the geometry.

3. In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.
4. In the **Settings** window for Gauge Fixing for A-Field, click to expand the **Advanced settings** section.

5 Locate the **Advanced Settings** section. Select the **Ensure constraint on value** check box.

Set up rotation of the rotor domain.

Prescribed Rotational Velocity 1

1. On the **Physics** toolbar, click **Domains** and choose **Prescribed Rotational Velocity**.
2. Select Domains 3 and 4 only.
3. In the **Settings** window for Prescribed Rotational Velocity, locate the **Prescribed Rotational Velocity** section.
4. In the rps text field, type `rpm`.

The scalar and vector potentials are connected via a special boundary condition, which is applied by default at the interface between the two formulations.

A continuity feature has to be added to specify the coupling across the pair. Note that pair features can be applied only if the same formulation is active on both sides of the pair.

Continuity 1

1. On the **Physics** toolbar, in the Boundary section, click **Pairs** and choose **Continuity**.
2. In the **Settings** window for Continuity, locate the **Pair Selection** section.
3. In the **Pairs** list, select **Identity Pair 1 (ap1)**.

The scalar potential also needs a point constraint, which is readily available as a standard point feature.

Zero Magnetic Scalar Potential 1

1. On the **Physics** toolbar, click **Points** and choose **Zero Magnetic Scalar Potential**.
2. Select Point 1 only.

MESH 1

Some extra care is needed for the meshing of source and destination boundaries for the pair; the destination side needs a finer mesh than the source side. To get full control, mesh these surfaces separately. Use a finer mesh for the copper domain to better resolve the expected velocity skin effect.

Free Triangular 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.
2. In the **Settings** window for Free Triangular, locate the **Boundary Selection** section.
3. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `11,12,14,15` in the **Selection** text field.

5 Click **OK**.

Size 1

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

2 In the **Settings** window for Size, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

5 In the associated text field, type `5e-3`.

Free Triangular 2

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `17,18,25,28` in the **Selection** text field.

5 Click **OK**.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 2** and choose **Size**.

2 In the **Settings** window for Size, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

5 In the associated text field, type `3e-3`.

Free Triangular 3

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `21, 22, 23, 24, 26, 27` in the **Selection** text field.

5 Click **OK**.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 3** and choose **Size**.

2 In the **Settings** window for Size, locate the **Element Size** section.

3 Click the **Custom** button.

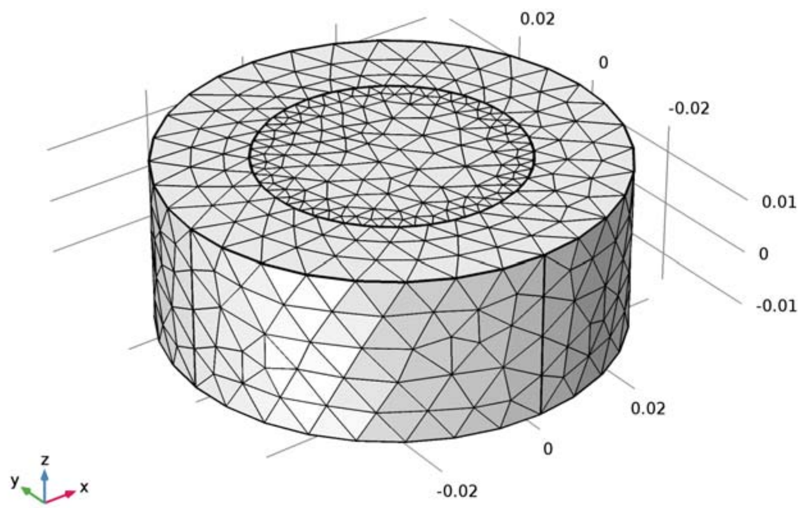
4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

5 In the associated text field, type `5e-3`.

Free Tetrahedral 1

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

2 In the **Settings** window for Free Tetrahedral, click **Build All**.



STUDY 1

Add and configure the time-dependent study and then solve. The stationary solution is automatically used as initial condition.

Time Dependent

On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

Step 2: Time Dependent

1 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

2 Click **Range**.

3 In the **Range** dialog box, choose **Number of values** from the Entry method list.

4 In the **Stop** text field, type `0.1`.

5 In the **Number of values** text field, type 31.

6 Click **Replace**.

7 On the **Study** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density (rmm)

1 In the **Model Builder** window, under **Results** click **Magnetic Flux Density (rmm)**.

2 On the **Magnetic Flux Density (rmm)** toolbar, click **Plot**.

Study 1/Solution 1 (sol1)

Change the frame of the data set. It needs to be spatial (the fixed "laboratory" frame) to see the rotor spin.

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Solution 1 (sol1)**.

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

3 From the **Frame** list, choose **Spatial (x, y, z)**.

Magnetic Flux Density (rmm)

1 In the **Model Builder** window, under **Results** click **Magnetic Flux Density (rmm)**.

2 On the **Magnetic Flux Density (rmm)** toolbar, click **Plot**.

Finish the modeling session by plotting the induced eddy currents in the copper disc.

3D Plot Group 2

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

For new plot groups, the data set edges are by default plotted on the material frame. Change to spatial frame.

2 In the **Settings** window for 3D Plot Group, locate the **Plot Settings** section.

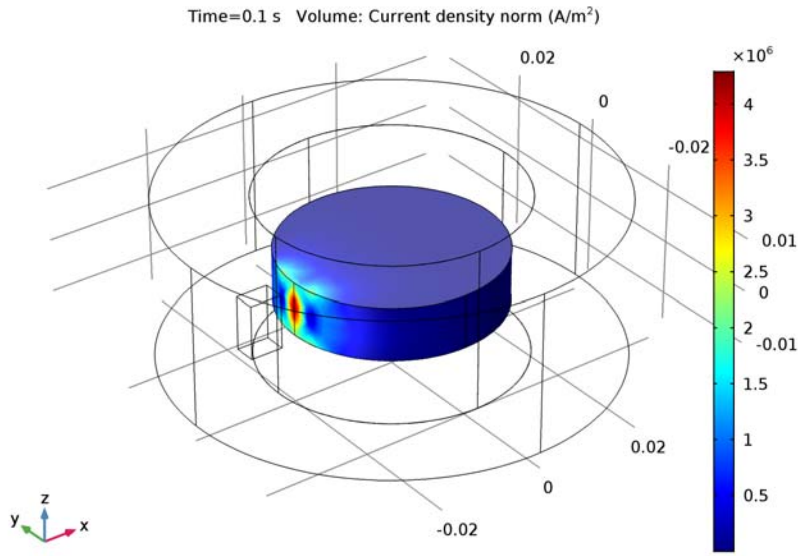
3 From the **Frame** list, choose **Spatial (x, y, z)**.

Volume 1

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

2 In the **Settings** window for Volume, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Rotating Machinery, Magnetic (Magnetic Fields)>Currents and charge>rmm.normJ - Current density norm**.

3 On the **3D Plot Group 2** toolbar, click **Plot**.



Created in COMSOL Multiphysics 5.2a

Computing the Resistance of a Wire

Introduction

Applying a voltage difference to a conductor creates a current flow and the intensity of the current is usually a function of the applied voltage difference. In the simplest (linear) case, the current flow and the voltage difference are proportional; the proportionality constant is the resistance of the device. This model demonstrates how to compute the resistance of a short section of copper wire. The convergence of the solution with respect to the mesh size is also studied.

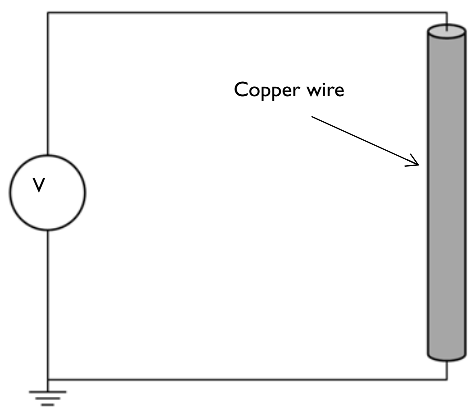


Figure 1: A short section of copper wire. The objective is to compute the equivalent resistance of this wire.

Model Definition

A 10 mm long section of copper wire of 1 mm radius, as shown in Figure 1, is studied. A constant current of 1 A is passed through the wire and the resistance of the wire is computed, as well as the voltage drop.

The boundary conditions used are meant to represent a connection to a DC source of current. One end of the wire is grounded, representing a current sink, and the other end is connected to a constant current source of 1 A, using the Terminal boundary condition.

Three different meshes are studied, to demonstrate that the results are converged with respect to mesh refinement—any further refinement of the mesh would only marginally improve the precision of the results. A Free Tetrahedral mesh is used, with varying default element sizes. The results are compared, and mesh convergence is shown.

Results and Discussion

The voltage distribution is plotted in Figure 2. A linear drop in the voltage along the length of the wire can be observed. The resistance of this 10 mm long wire is computed to be 0.212 mΩ., a value that agrees within 1% for all meshes.

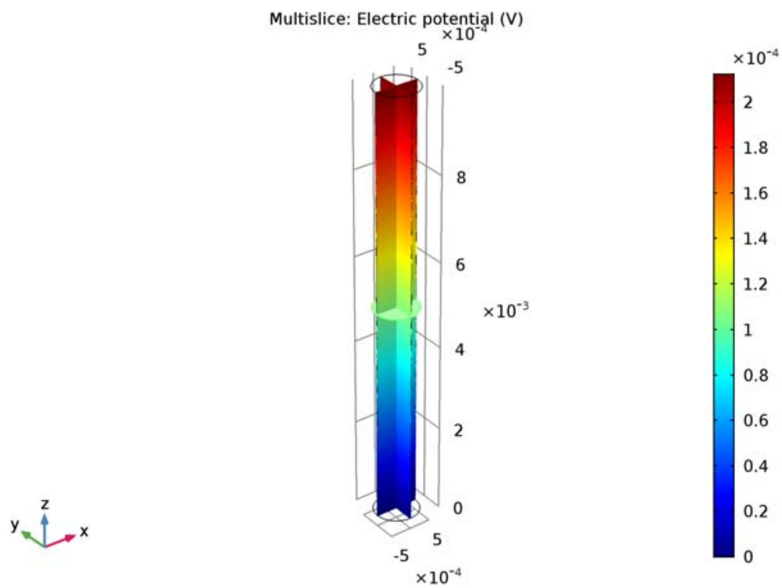


Figure 2: The voltage decreases linearly along the length of the wire.

Application Library path: `ACDC_Module/Resistive_Devices/simple_resistor`

Modeling Instructions

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

NEW

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

MODEL WIZARD

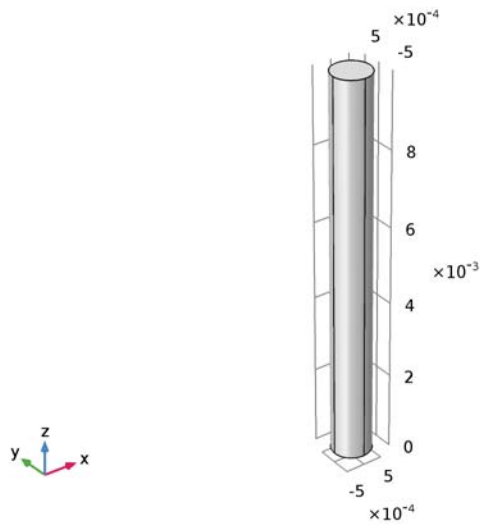
1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **AC/DC>Electric Currents (ec)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Stationary**.
6 Click **Done**.

GEOMETRY 1

Begin by creating a cylinder for the copper wire.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for Cylinder, locate the **Size and Shape** section.
3 In the **Radius** text field, type `0.5[mm]`.
4 In the **Height** text field, type `10[mm]`.
5 Click **Build All Objects**.



6 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

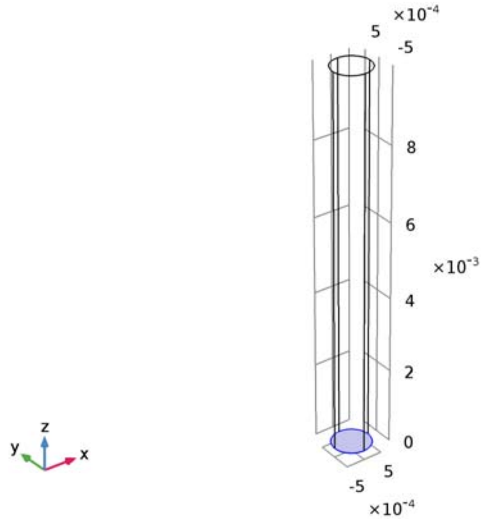
ELECTRIC CURRENTS (EC)

Set up the **Electric Current** physics. Specify the ground and terminal boundaries.

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

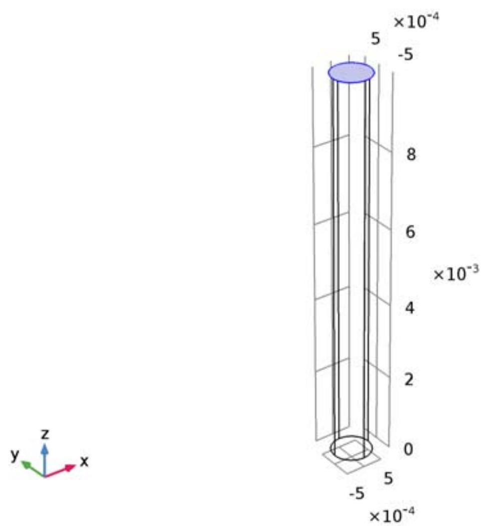
2 Select Boundary 3 only.



Terminal 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Terminal**.

2 Select Boundary 4 only.



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

4 In the I_0 text field, type 1.

MATERIALS

Then, assign material properties. Use copper for all domains.

ADD MATERIAL

1 On 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.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MESH 1

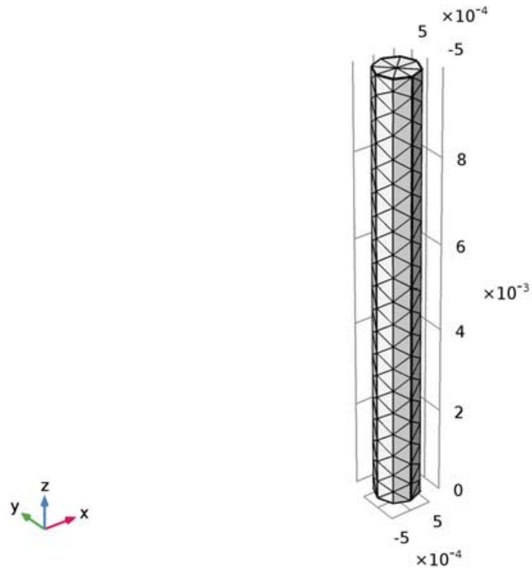
In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

Size

1 In the **Settings** window for Size, locate the **Element Size** section.

2 From the **Predefined** list, choose **Extra coarse**.

3 Click **Build All**.



STUDY 1

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

RESULTS

Electric Potential (ec)

The default plot shows the electric potential in the copper wire. See Figure 2.

Derived Values

Evaluate the resistance of the wire with the extra coarse mesh size.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for Global Evaluation, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Model>Component 1>Electric Currents>Terminals>ec.R11 - Resistance**.

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

Expression	Unit	Description
ec.R11	mΩ	Resistance

4 Click **Evaluate**.

MESH 1

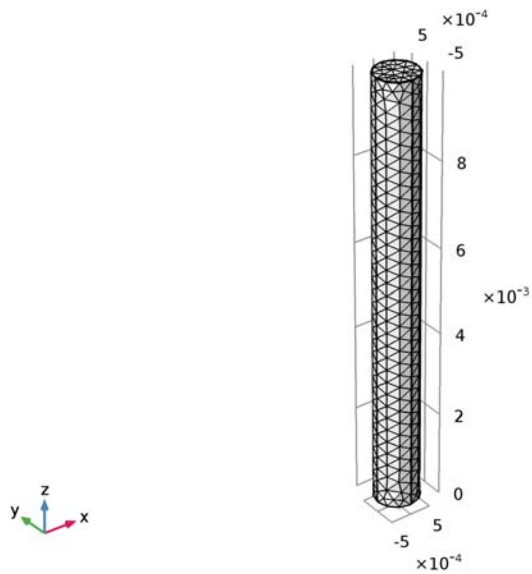
Size

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

2 In the **Settings** window for Size, locate the **Element Size** section.

3 From the **Predefined** list, choose **Normal**.

4 Click **Build All**.



STUDY 1

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

RESULTS

Global Evaluation 1

Evaluate the resistance of the wire with the normal mesh size.

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 1**.

2 In the **Settings** window for Global Evaluation, click **Evaluate**.

MESH 1

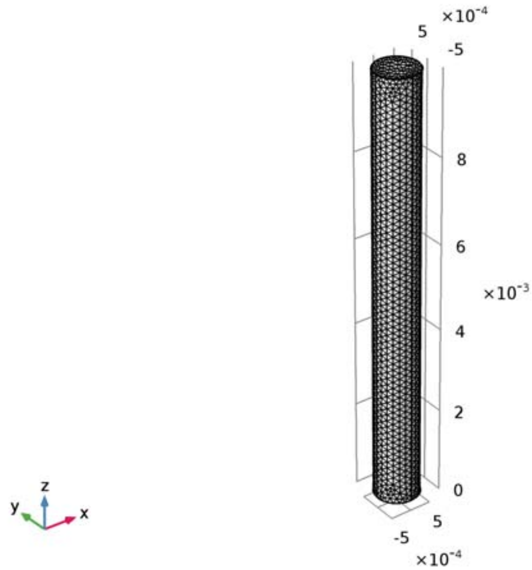
Size

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

2 In the **Settings** window for Size, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

4 Click **Build All**.



STUDY 1

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

RESULTS

Global Evaluation 1

Finish the result analysis by evaluating the resistance of the wire with the extra fine mesh size.

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 1**.

2 In the **Settings** window for Global Evaluation, click **Evaluate**.

TABLE

1 Go to the **Table** window.

The evaluated wire resistance for the three different meshes should agree within 1%.

Created in COMSOL Multiphysics 5.2a

Small-Signal Analysis of an Inductor

Introduction

This example uses the model Inductor in an Amplifier Circuit from the *AC/DC Module Application Library* without the circuits definition. This model consists of an inductor with a nonlinear magnetic core that shows a changing inductance when the current increases. In this example you investigate the small-signal inductance as a function of current through the inductor.

Application Library path: `ACDC_Module/Inductive_Devices_and_Coils/small_signal_analysis_of_inductor`

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 **2D Axisymmetric**.

2 In the **Select Physics** tree, select **AC/DC>Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Small-Signal Analysis, Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
dr	3[cm]	0.03 m	Domain Radius
il	2[cm]	0.02 m	Inductor Length

Name	Expression	Value	Description
cr	5[mm]	0.005 m	Core Radius
fr	0.5[mm]	5E-4 m	Fillet Radius
coor	10.5[mm]	0.0105 m	Coil, Outer Radius
coir	7.5[mm]	0.0075 m	Coil, Inner Radius
cwc	pi*(0.05[mm])^2	7.854E-9 m²	Coil, Wire Cross Section
cn	1000	1000	Coil, Number of Turns
f0	10[kHz]	10000 Hz	Operating Frequency
csigma	5e7[S/m]	5E7 S/m	Coil, Wires Conductivity
cIdc	1[A]	1 A	DC Current Bias
cIac	1[A]	1 A	AC Current Perturbation

GEOMETRY 1

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `dr`.

4 Right-click **Circle 1 (c1)** and choose **Build Selected**.

The part of the circle left of the axis will be automatically removed when the geometry is finalized.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `cr`.

4 In the **Height** text field, type `il`.

5 Locate the **Position** section. In the **z** text field, type `-il/2`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `coor-coir`.

4 In the **Height** text field, type `il`.

5 Locate the **Position** section. In the **r** text field, type `coir`.

6 In the **z** text field, type `-il/2`.

Fillet 1 (fil1)

1 On the **Geometry** toolbar, click **Fillet**.

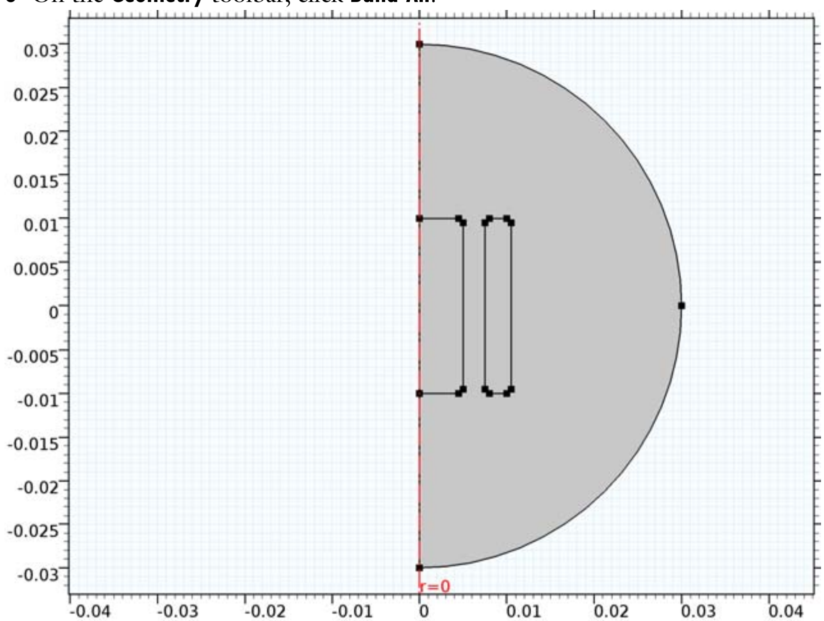
2 On the object **r2**, select Points 1–4 only.

3 On the object **r1**, select Points 2 and 3 only.

4 In the **Settings** window for Fillet, locate the **Radius** section.

5 In the **Radius** text field, type `fr`.

6 On the **Geometry** toolbar, click **Build All**.



This is the final geometry.

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

ADD MATERIAL

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

2 In the tree, select **AC/DC>Soft Iron (without losses)**.

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

MATERIALS

Soft Iron (without losses) (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Soft Iron (without losses) (mat2)**.

2 Select Domain 2 only.

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

MAGNETIC FIELDS (MF)

The nonlinear inductor must use a different constitutive relation for the magnetic field. Thus a separate **Ampère's Law** feature must be entered for this.

Ampère's Law 2

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 In the **Model Builder** window, right-click **Ampère's Law 2** and choose **Rename**.

3 In the **Rename Ampère's Law** dialog box, type `Nonlinear Core` in the **New label** text field.

4 Click **OK**.

5 Select Domain 2 only.

6 In the **Settings** window for Ampère's Law, locate the **Magnetic Field** section.

7 From the **Constitutive relation** list, choose **HB curve**.

Define the coil with the static excitation.

Coil 1

1 On the **Physics** toolbar, click **Domains** and choose **Coil**.

2 Select Domain 3 only.

3 In the **Settings** window for Coil, locate the **Coil** section.

4 In the **Coil name** text field, type `coil`.

5 From the **Conductor model** list, choose **Homogenized multi-turn**.

6 Locate the **Homogenized Multi-Turn Conductor** section. In the N text field, type `cn`.

7 In the σ_{coil} text field, type `csigma`.

8 In the a_{coil} text field, type `cwc`.

9 Locate the **Coil** section. In the I_{coil} text field, type `cIdc`.

Add the time harmonic excitation.

Harmonic Perturbation 1

1 On the **Physics** toolbar, click **Attributes** and choose **Harmonic Perturbation**.

2 In the **Settings** window for Harmonic Perturbation, locate the **Harmonic Perturbation** section.

3 In the I_{coil} text field, type `cIac`.

DEFINITIONS

Define a scalar estimate of the relative permittivity. It will be plotted to show the saturation of the iron core.

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
mur	mf.normB/(mu0_const* mf.normH)		Relative Permeability

STUDY 1

The stationary solver will sweep the value of the parameter `cIdc` (DC bias current) over a range from 1 A to 10 A. The stationary solution computed at each point will be used as linearization point for the corresponding frequency-domain perturbation step. Thus you need to set up a continuation sweep in the stationary solver.

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

2 In the **Settings** window for Stationary, click to expand the **Study extensions** section.

3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
cIdc	range(1,0.2,10)	

Step 2: Frequency-Domain Perturbation

1. In the **Model Builder** window, under **Study 1** click **Step 2: Frequency-Domain Perturbation**.
2. In the **Settings** window for Frequency-Domain Perturbation, locate the **Study Settings** section.
3. In the **Frequencies** text field, type `f0`.

   Apply to the frequency-domain solver the same sweep that was applied to the stationary solver. This ensures that the stationary parameter (the DC current) is recognized properly.
4. Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
5. Click **Add**.
6. In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
cIdc	`range(1,0.2,10)`	

Solution stability benefits from tightening the nonlinear tolerance for the stationary step.

Solution 1 (sol1)

1. On the **Study** toolbar, click **Show Default Solver**.
2. In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.
3. In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Stationary Solver 1**.
4. In the **Settings** window for Stationary Solver, locate the **General** section.
5. In the **Relative tolerance** text field, type `1e-6`.
6. On the **Study** toolbar, click **Compute**.

RESULTS

1D Plot Group 3

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

Global 1

1. On the **1D Plot Group 3** toolbar, click **Global**.

2 In the **Settings** window for Global, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Magnetic Fields>Coil parameters>mf.LCoil_coil - Coil inductance**.

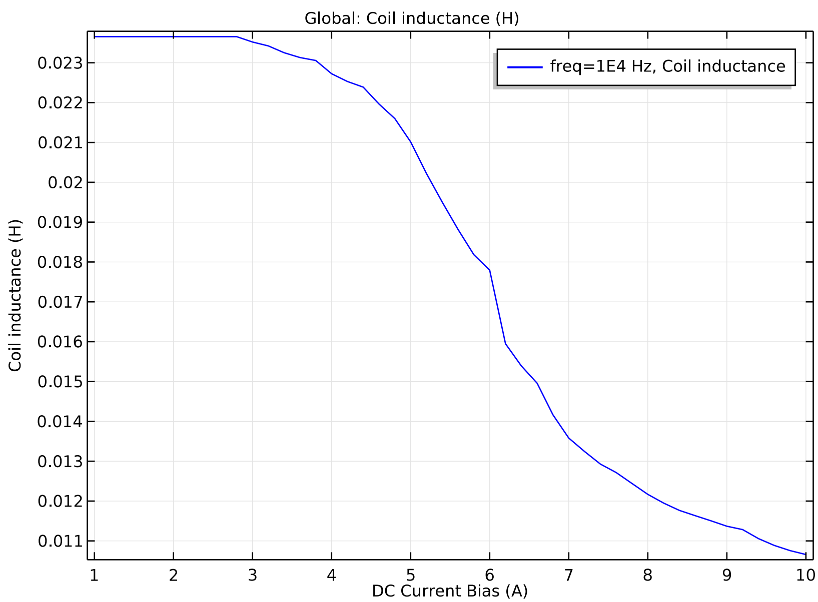
The Global Plot can be used to plot physical quantities from the stationary solution, from the harmonic perturbation solution, or in other cases available in the **Expression evaluated for** list. The **Compute differential** check box is used to compute the differential of the physical quantity around the linearization point. In the case of lumped parameters, the differential should not be computed, since they are defined as the ratio of two differentials. Leave that option unchecked.

3 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type `cIdc`.

5 On the **1D Plot Group 3** toolbar, click **Plot**.

The plot will show the inductance of the component at different values of the bias currents.The inductance drops to zero as a consequence of the saturation of the core. The saturation can be visualized by plotting the relative permeability of the iron core.



The small signal inductance is plotted versus the DC bias current.

Magnetic Flux Density Norm (mf)

1 In the **Model Builder** window, under **Results** click **Magnetic Flux Density Norm (mf)**.

2 In the **Settings** window for 2D Plot Group, locate the **Data** section.

3 From the **Parameter value (cldc)** list, choose **5.8**.

Surface 1

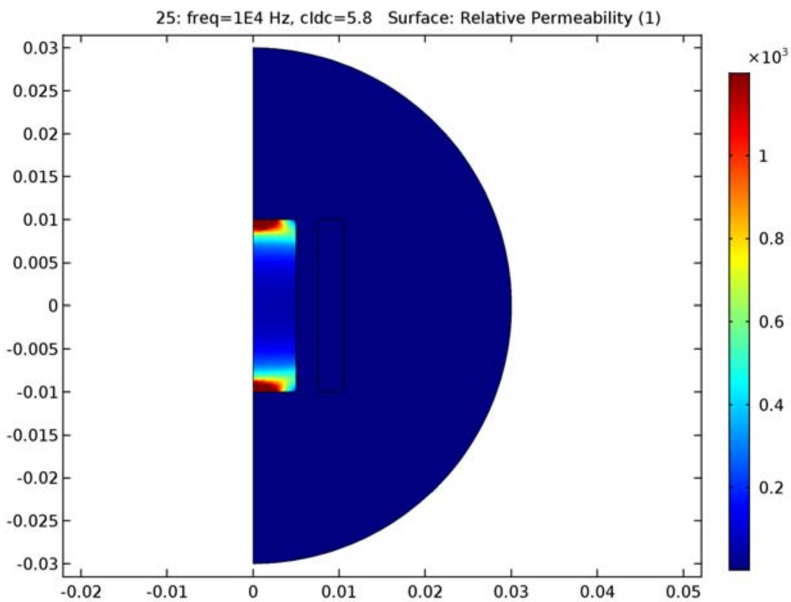
1 In the **Model Builder** window, expand the **Magnetic Flux Density Norm (mf)** node, then click **Surface 1**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Definitions> Variables>mur - Relative Permeability**.

3 Locate the **Expression** section. From the **Expression evaluated for** list, choose **Static solution**.

4 On the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

The plot shows that the relative permeability almost everywhere has dropped well below its maximum value of 1200.



The Harmonic Perturbation subfeature can be added to a wide range of source features in the AC/DC module. A linear perturbation study can be manually performed on any other source by using the linper operator as shown next.

MAGNETIC FIELDS (MF)

Coil 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Fields (mf)** click **Coil 1**.

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

3 In the I_{coil} text field, type `cIdc + linper(cIac)`.

Harmonic Perturbation 1

In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Fields (mf)>Coil 1** right-click **Harmonic Perturbation 1** and choose **Disable**.

STUDY 1

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

RESULTS

Magnetic Flux Density Norm (mf)

Inspect the results again and confirm they did not change.

Created in COMSOL Multiphysics 5.2a

Static Field Modeling of a Halbach Rotor

Introduction

This example presents the static-field modeling of an outward-flux-focusing magnetic rotor using permanent magnets, a magnetic rotor also known as a Halbach rotor. The use of permanent magnets in rotatory devices such as motors, generators, and magnetic gears is increasing due to their no-contact, frictionless operation. This model illustrates how to calculate the magnetic field of a 4-pole pair rotor in 3D by modeling only a single pole of the rotor using symmetry.

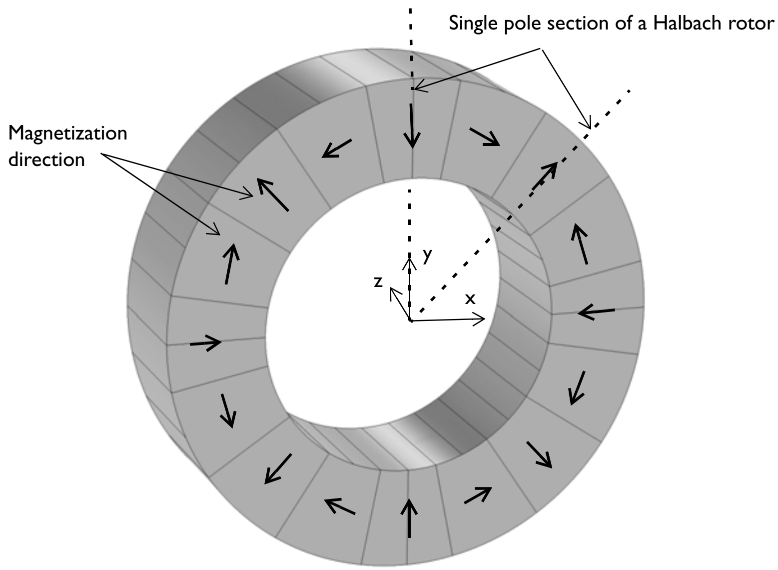


Figure 1: Illustration of a 16-segments, 4-pole pair Halbach rotor. The symmetries of the problem allow restricting the model to a single pole of the rotor.

Model Definition

Set up the problem in a 3D modeling space. Due to symmetry, it is sufficient to model a single pole of the rotor. Figure 1 shows a 3D view of the complete rotor with the magnetization direction of the magnets indicated. The black arrows show the radial and axial magnetization directions of the permanent magnets in the rotor. The permanent magnets are arranged in such a way that the magnetic flux density is minimized inside the rotor and maximized outside the rotor. The model consists of 16 permanent magnet pieces arranged to form a 4-pole pair rotor. The inner and outer rotor radii are 30 mm and 50 mm, respectively. The axial length of the rotor is 30 mm.

Results and Discussion

A steady-state study analysis is performed to calculate the magnetic fields of the Halbach rotor. The magnetic flux density is shown in Figure 2.

Figure 3 and Figure 4 illustrate the variations of the radial and azimuthal magnetic flux density as functions of rotor angle. The magnetic flux density norm is evaluated outside the Halbach rotor at a radial distance of 55 mm from the center.

Finally, Figure 5 and Figure 6 show the polar plots of the magnetic flux density norm at radial distances from the rotor center of 55 mm and 25 mm, respectively.

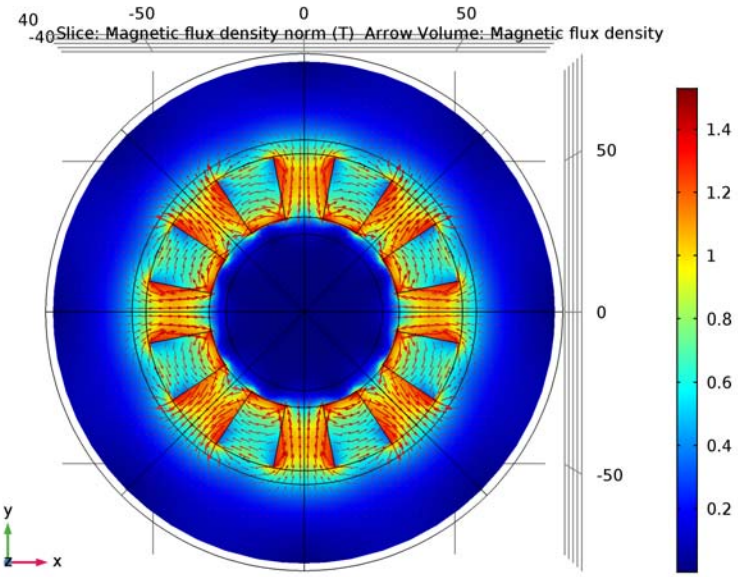


Figure 2: Magnetic flux density norm at the cross section of the Halbach rotor (at $z = 0$ mm).

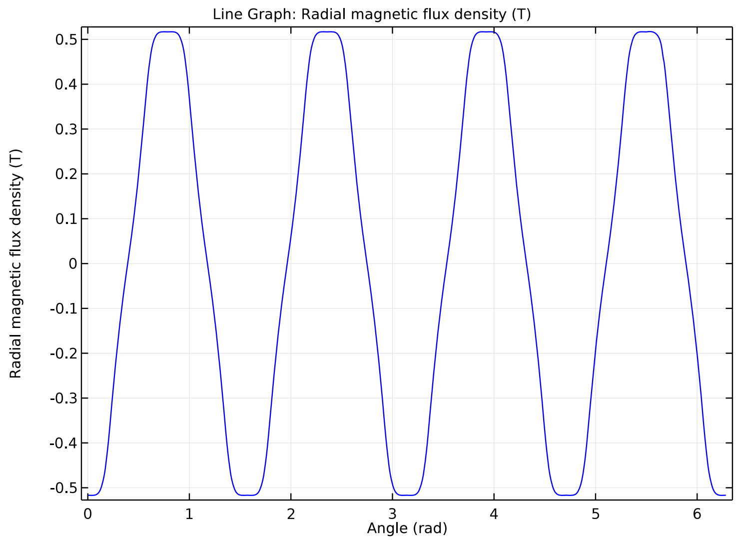


Figure 3: The radial magnetic flux density as a function of rotor angle measured at a radial distance of 55 mm from the rotor center.

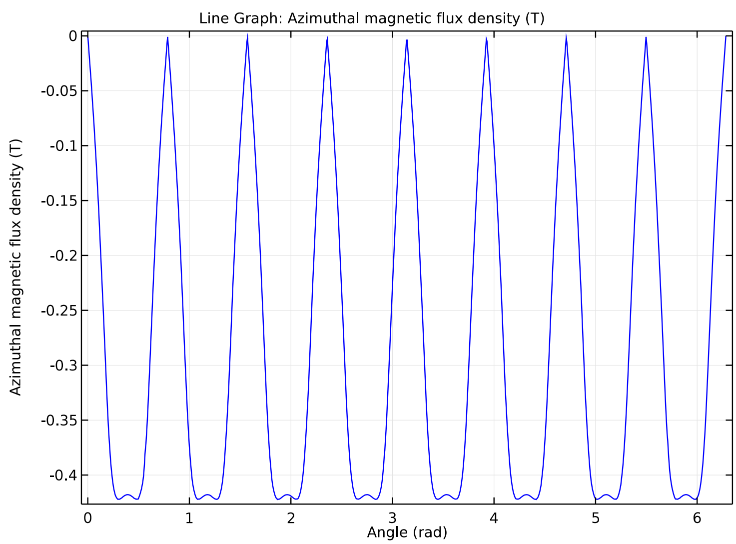


Figure 4: The azimuthal magnetic flux density as a function of rotor angle measured at a radial distance of 25 mm from the rotor center.

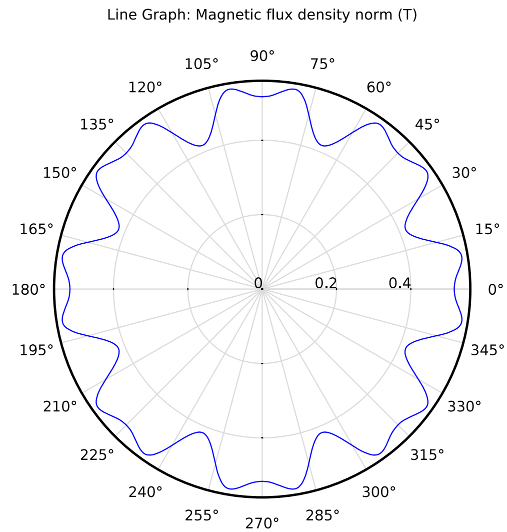


Figure 5: Polar plot of the magnetic flux density norm at a radial distance of 55 mm from the rotor center.

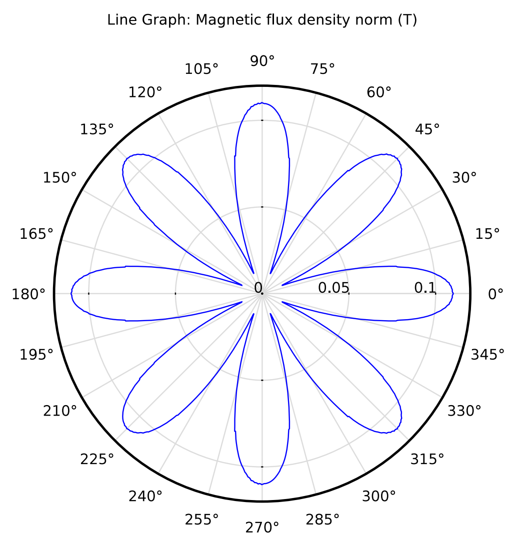


Figure 6: Polar plot of the magnetic flux density norm at a radial distance of 25 mm from the rotor center.

Application Library path: `ACDC_Module/Magnetostatics/static_field_halbach_rotor_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>Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Define all the model parameters here.

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
Ro	50[mm]	0.05 m	Outer radius of the rotor
Ri	30[mm]	0.03 m	Inner radius of the rotor
L	30[mm]	0.03 m	Length of the rotor
alpha	11.25[deg]	0.1963 rad	Angle of rotation

GEOMETRY 1

Insert the geometry sequence from the `static_field_halbach_rotor_3d_geom_sequence.mph` file.

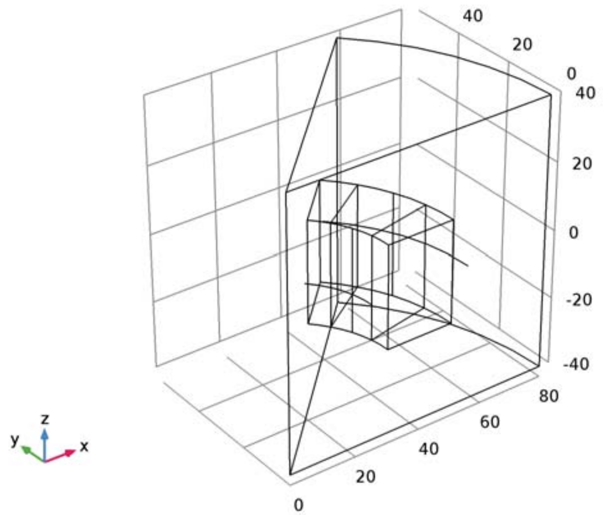
1 On the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the application's Application Libraries folder and double-click the file `static_field_halbach_rotor_3d_geom_sequence.mph`.

Form Union (fin)

1 On the **Geometry** toolbar, click **Build All**.

2 Click the **Wireframe Rendering** button on the **Graphics** toolbar.



DEFINITIONS

Define variables for the radial and azimuthal magnetic flux densities.

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
R	sqrt(x^2+y^2)	m	Radial distance
B_r	(mf.Bx*x+mf.By*y)/R	T	Radial magnetic flux density
B_phi	(-mf.Bx*y+mf.By*x)/R	T	Azimuthal magnetic flux density

Define a selection for the magnets.

Explicit 1

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

2 Select Domains 2–4 only.

3 Right-click **Explicit 1** and choose **Rename**.

4 In the **Rename Explicit** dialog box, type `Magnets` in the **New label** text field.

5 Click **OK**.

Add a new cylindrical coordinate system. You will use this coordinate system to assign the magnetization of the permanent magnets.

Cylindrical System 2 (sys2)

On the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.

View 1

Hide a few boundaries to view the results only in the inner part of the model domain.

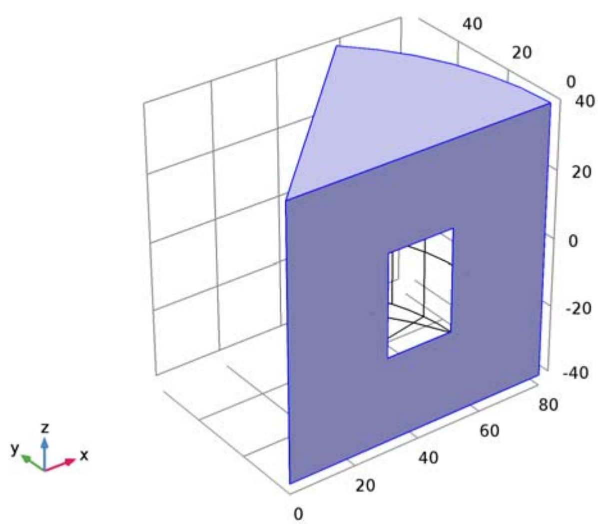
Hide for Physics 1

1 On the **View 1** toolbar, click **Hide for Physics**.

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

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

4 Select Boundaries 1, 2, and 4 only.



MAGNETIC FIELDS (MF)

Now, set up the **Magnetic Fields** physics. Model the permanent magnets using **Ampère's Law**.

Ampère's Law 2

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domain 2 only.

3 In the **Settings** window for Ampère's Law, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

5 Locate the **Magnetic Field** section. From the **Constitutive relation** list, choose **Remanent flux density**.

6 Specify the $\mathbf{B}_\mathrm{r}$ vector as

1.42	r
0	phi
0	a

Ampère's Law 3

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domain 4 only.

3 In the **Settings** window for Ampère's Law, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

5 Locate the **Magnetic Field** section. From the **Constitutive relation** list, choose **Remanent flux density**.

6 Specify the $\mathbf{B}_\mathrm{r}$ vector as

-1.42	r
0	phi
0	a

Ampère's Law 4

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domain 3 only.

3 In the **Settings** window for Ampère's Law, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

5 Locate the **Magnetic Field** section. From the **Constitutive relation** list, choose **Remanent flux density**.

6 Specify the $\mathbf{B_r}$ vector as

0	r
1.42	phi
0	a

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MESH 1

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

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

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

Size 1

1 Right-click **Component 1 (comp1)>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 **Magnets**.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Fine**.

Specify a very fine mesh on the curves where the magnetic flux density is to be evaluated. This helps to obtain a smooth curve for magnetic flux density.

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 **Edge**.

4 Select Edges 6 and 31 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.

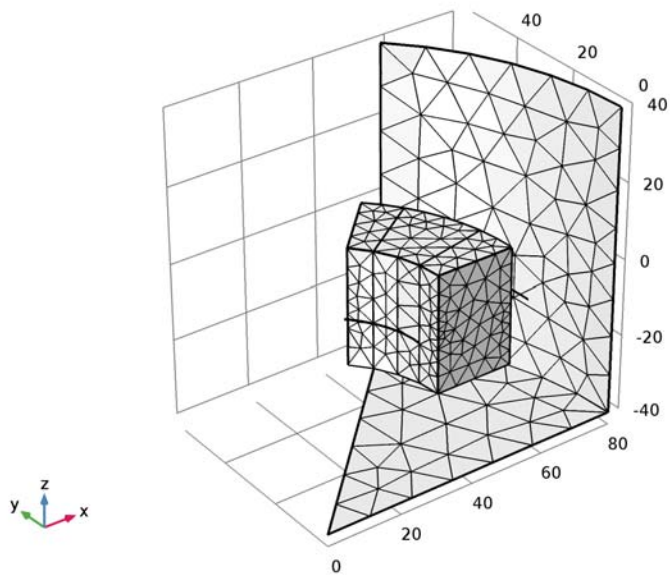
7 In the associated text field, type `0.5`.

Free Tetrahedral 1

1 Right-click **Mesh 1** and choose **Free Tetrahedral**.

2 In the **Settings** window for Free Tetrahedral, click **Build All**.

Compare the mesh with the figure shown below.



STUDY 1

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

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

3 On the **Home** toolbar, click **Compute**.

RESULTS

Data Sets

Use the Sector 3D data set to produce a 3D data set for the complete 3D model from the single-pole results.

Sector 3D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Sector 3D**.

2 In the **Settings** window for Sector 3D, locate the **Symmetry** section.

3 In the **Number of sectors** text field, type 8.

4 From the **Transformation** list, choose **Rotation and reflection**.

Construct circles inside and outside of the Halbach rotor to visualize the magnetic flux density along this curve.

Parameterized Curve 3D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Parameterized Curve 3D**.

2 In the **Settings** window for Parameterized Curve 3D, locate the **Data** section.

3 From the **Data set** list, choose **Sector 3D 1**.

4 Locate the **Parameter** section. In the **Name** text field, type `phi`.

5 In the **Maximum** text field, type `2*pi`.

6 Locate the **Expressions** section. In the **x** text field, type `55*cos(phi)`.

7 In the **y** text field, type `55*sin(phi)`.

Parameterized Curve 3D 2

1 On the **Results** toolbar, click **More Data Sets** and choose **Parameterized Curve 3D**.

2 In the **Settings** window for Parameterized Curve 3D, locate the **Data** section.

3 From the **Data set** list, choose **Sector 3D 1**.

4 Locate the **Parameter** section. In the **Name** text field, type `phi`.

5 In the **Maximum** text field, type `2*pi`.

6 Locate the **Expressions** section. In the **x** text field, type `25*cos(phi)`.

7 In the **y** text field, type `25*sin(phi)`.

Use the following instructions to reproduce the plot shown in Figure 2.

3D Plot Group 1

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for 3D Plot Group, locate the **Data** section.

3 From the **Data set** list, choose **Sector 3D 1**.

Slice 1

1 Right-click **3D Plot Group 1** and choose **Slice**.

2 In the **Settings** window for Slice, locate the **Plane Data** section.

3 From the **Plane** list, choose **xy-planes**.

4 In the **Planes** text field, type `1`.

5 On the **3D Plot Group 1** toolbar, click **Plot**.

3D Plot Group 1

In the **Model Builder** window, under **Results** right-click **3D Plot Group 1** and choose **Arrow Volume**.

Arrow Volume 1

1 In the **Settings** window for Arrow Volume, locate the **Arrow Positioning** section.
2 Find the **x grid points** subsection. In the **Points** text field, type `60`.
3 Find the **y grid points** subsection. In the **Points** text field, type `60`.
4 Find the **z grid points** subsection. In the **Points** text field, type `1`.
5 On the **3D Plot Group 1** toolbar, click **Plot**.
6 Click the **Go to XY View** button on the **Graphics** toolbar.

Next, generate a plot for the radial magnetic flux density outside the Halbach rotor. Compare the result with Figure 3.

1D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for 1D Plot Group, locate the **Data** section.
3 From the **Data set** list, choose **Parameterized Curve 3D 1**.
4 Locate the **Plot Settings** section. Select the **x-axis label** check box.
5 In the associated text field, type `Angle (rad)`.

Line Graph 1

1 On the **1D Plot Group 2** toolbar, click **Line Graph**.
2 In the **Settings** window for Line Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables>B_r - Radial magnetic flux density**.
3 On the **1D Plot Group 2** toolbar, click **Plot**.

Create the azimuthal magnetic flux density plot as shown in Figure 4.

1D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for 1D Plot Group, locate the **Data** section.
3 From the **Data set** list, choose **Parameterized Curve 3D 1**.
4 Locate the **Plot Settings** section. Select the **x-axis label** check box.
5 In the associated text field, type `Angle (rad)`.

Line Graph 1

1 On the **1D Plot Group 3** toolbar, click **Line Graph**.

2 In the **Settings** window for Line Graph, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables>B_phi - Azimuthal magnetic flux density**.

3 On the **1D Plot Group 3** toolbar, click **Plot**.

Generate the polar plot of the magnetic flux density norm at a distance 55 mm away from the center of the rotor.

Polar Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **Polar Plot Group**.

2 In the **Settings** window for Polar Plot Group, locate the **Data** section.

3 From the **Data set** list, choose **Parameterized Curve 3D 1**.

4 Click to expand the **Axis** section. Select the **Manual axis limits** check box.

5 In the **r maximum** text field, type `0.56`.

Line Graph 1

1 On the **Polar Plot Group 4** toolbar, click **Line Graph**.

2 Click **Plot**.

Finally, reproduce the plot for the magnetic flux density norm at a distance 25 mm from the rotor center.

Polar Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **Polar Plot Group**.

2 In the **Settings** window for Polar Plot Group, locate the **Data** section.

3 From the **Data set** list, choose **Parameterized Curve 3D 2**.

4 Locate the **Axis** section. Select the **Manual axis limits** check box.

5 In the **r maximum** text field, type `0.12`.

Line Graph 1

1 On the **Polar Plot Group 5** toolbar, click **Line Graph**.

2 Click **Plot**.

Compare this figure with that shown in Figure 6.

Magnetic Signature of a Submarine

Introduction

A vessel traveling on the surface or under water gives rise to detectable local disturbances in the Earth's magnetic field. These disturbances can be used to trigger weapon systems. The magnetic signature of a ship can be reduced by generating a counteracting magnetic field of suitable strength and direction based on prior knowledge of the magnetic properties of the vessel. An important step in the design of a naval ship is therefore to predict its magnetic signature. Another application where magnetic signatures are of great importance is in urban traffic control: magnetic sensors, buried in our streets, are used to sense vehicles and control traffic lights.

Ships and cars are both to a large extent made of sheet metal. This makes them hard to simulate using standard finite element analysis because volume meshes of thin extended structures are difficult to generate and tend to become very large. This application demonstrates a powerful technique that circumvents the problem by modeling the sheet metal as 2D faces embedded in a 3D geometry. Thus it is only necessary to create comparatively inexpensive 2D face meshes in addition to the 3D volume mesh used for the surrounding medium. A tangential projection of the 3D equation is then solved on the 2D face mesh.

Figure 1: Submarine HMAS Collins. Image courtesy of Kockums AB.

This application also demonstrates the use of the *reduced field formulation* available in the AC/DC Module. This feature provides a convenient way to obtain the magnetic signature of the submarine by allowing the user to define the background field as a predefined quantity and solving only for the perturbations in this field.

Model Definition

In magnetostatic problems, where no currents are present, the problem can be solved using a scalar magnetic potential. This application demonstrates a special technique for modeling thin sheets of high permeability materials, and also shows the use of the reduced field formulation in the AC/DC Module for conveniently modeling perturbations in a known background field.

The model geometry is shown in Figure 2 and consists of face objects representing the submarine. A 3D box representing the surrounding water encloses the vessel.

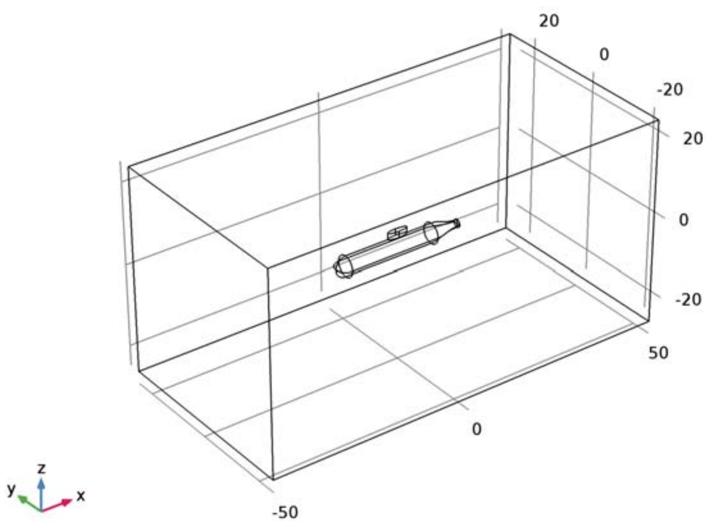


Figure 2: The model geometry.

DOMAIN EQUATIONS

In a current-free region, where

$$\nabla \times \mathbf{H} = \mathbf{0}$$

it is possible to define the scalar magnetic potential, V_{m}, from the relation

$$\mathbf{H} = -\nabla V_{\mathrm{m}}$$

This is analogous to the definition of the electric potential for static electric fields. Using the constitutive relation between the magnetic flux density and magnetic field

$$\mathbf{B} = \mu_0 \mu_r \mathbf{H}$$

together with the equation

$$\nabla \cdot \mathbf{B} = 0$$

you can derive an equation for V_m,

$$-\nabla \cdot (\mu_0 \mu_r \nabla V_m) = 0$$

In this model you use the reduced field formulation, which means that you only solve for the potential V_m corresponding to the perturbation (reduced) field, so the equation you solve reads

$$-\nabla \cdot (\mu_0 \mu_r \nabla V_m + \mathbf{B}_{ext}) = 0$$

where $\mathbf{B}_{ext}$ is a known background field, in this case the Earth's magnetic field of 0.5 G.

Boundary Conditions

The exterior boundaries of the box are insulating for the reduced magnetic field:

$$-\mathbf{n} \cdot \mathbf{B}_{red} = 0$$

On the face objects representing the hull of the submarine, you apply a 2D tangential projection of the 3D domain equation where the thickness and permeability of the hull are introduced as parameters. This is readily available in the used formulation as a shielding boundary condition, which is useful for modeling of highly permeable thin sheets. Corresponding boundary conditions are available in the Electric Currents and Electrostatics interfaces for modeling of thin sheets with high conductance and high permittivity respectively.

Results and Discussion

Figure 3 shows the total magnetic flux density in a horizontal slice plot 7.5 m below the keel of the submarine. A distinct field perturbation due to the presence of the vessel can be seen. The magnitude and direction of the tangential magnetic field in the hull of the vessel is shown using arrows. The reduced field is visualized as isolevels of the reduced magnetic potential. This gives a good picture of the perturbation caused by the presence of the submarine in the background field.

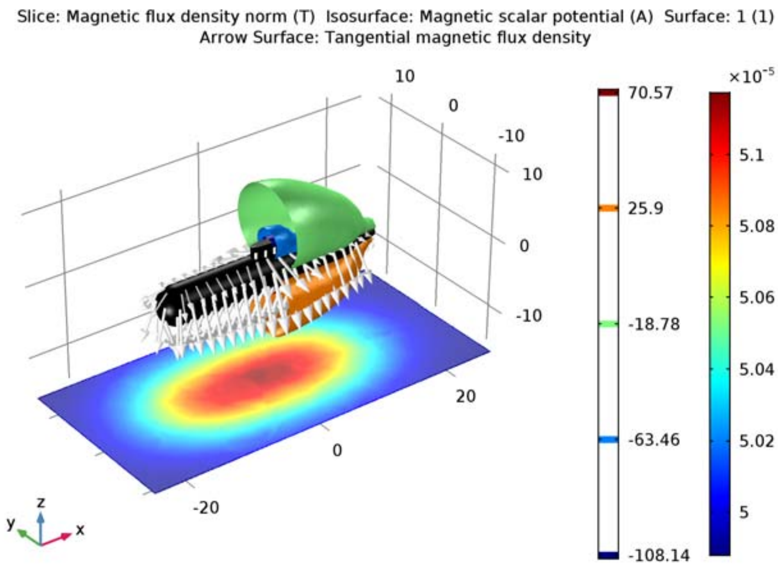


Figure 3: The slice color plot shows the total magnetic flux density (right color legend). The arrows show the direction and strength of the tangential magnetic field in the hull. The isolevels show the reduced magnetic potential (left color legend).

Application Library path: `ACDC_Module/Magnetostatics/submarine`

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>Magnetic Fields, No Currents (mfnc)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Load the dimensional parameters used when building the submarine geometry as well as a parameter for the strength of the geomagnetic field from a text file.

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 Click **Load from File**.

4 Browse to the application's Application Libraries folder and double-click the file `submarine.txt`.

GEOMETRY 1

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

1 On the **Geometry** toolbar, click **Insert Sequence**.

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

Form Union (fin)

On the **Geometry** toolbar, click **Build All**.

DEFINITIONS

View 1

Hide the boundaries of the water domain to show only the submarine.

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

Hide for Physics 1

1 On the **View 1** toolbar, click **Hide for Physics**.

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

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

4 Select Boundaries 1–5 and 27 only.

MAGNETIC FIELDS, NO CURRENTS (MFNC)

Apply a background magnetic field corresponding to Earth's geomagnetic field.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields, No Currents (mfnc)**.

2 In the **Settings** window for Magnetic Fields, No Currents, locate the **Background Magnetic Field** section.

3 From the **Solve for** list, choose **Reduced field**.

4 Specify the $\mathbf{H}_b$ vector as

0	x
0	y
gB/mu0_const	z

Magnetic Flux Conservation 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Fields, No Currents (mfnc)** click **Magnetic Flux Conservation 1**.

2 In the **Settings** window for Magnetic Flux Conservation, locate the **Magnetic Field** section.

3 From the μ_r list, choose **User defined**. The **External Magnetic Flux Density** feature imposes boundary conditions matching the specified background field.

External Magnetic Flux Density 1

1 On the **Physics** toolbar, click **Boundaries** and choose **External Magnetic Flux Density**.

2 In the **Settings** window for External Magnetic Flux Density, locate the **Boundary Selection** section.

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

The **Magnetic Shielding** feature models a thin layer of high permittivity material, such as the metal constituting the submarine hull.

Magnetic Shielding 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Magnetic Shielding**.

Select all the boundaries forming the submarine hull using the Select Box tool.

2 Click the **Select Box** button on the **Graphics** toolbar.

3 Select Boundaries 6–26 only.

4 In the **Settings** window for Magnetic Shielding, locate the **Magnetic Shielding** section.

5 From the μ_r list, choose **User defined**. In the associated text field, type `700`.

6 In the d_s text field, type `0.05`.

MESH 1

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

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

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

Size

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

2 In the **Settings** window for Size, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `5`.

5 In the **Curvature factor** text field, type `0.4`.

These settings will adequately resolve the region around the submarine hull.

6 Click **Build All**.

STUDY 1

Disable the automatic generation of default plots. Instead, you will create a custom plot when the solver has finished.

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

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

3 On the **Home** toolbar, click **Compute**.

RESULTS

3D Plot Group 1

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

2 In the **Settings** window for 3D Plot Group, locate the **Plot Settings** section.

3 Clear the **Plot data set edges** check box.

Slice 1

1 Right-click **3D Plot Group 1** and choose **Slice**.

2 In the **Settings** window for Slice, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields, No Currents>Magnetic>mfnc.normB - Magnetic flux density norm**.

3 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

4 From the **Entry method** list, choose **Coordinates**.

5 In the **z-coordinates** text field, type `-15`.

Add a filter to plot the slice only in proximity of the submarine.

Filter 1

1 Right-click **Results>3D Plot Group 1>Slice 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `abs(x)<(tl*.8)&&abs(y)<(r*5)`.

3D Plot Group 1

In the **Model Builder** window, under **Results** right-click **3D Plot Group 1** and choose **Isosurface**.

Isosurface 1

Add another filter to cut the isosurfaces and show the submarine hull inside.

Filter 1

1 In the **Model Builder** window, under **Results>3D Plot Group 1** right-click **Isosurface 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `x>0`.

3D Plot Group 1

In the **Model Builder** window, under **Results** right-click **3D Plot Group 1** and choose **Surface**.

Surface 1

1 In the **Settings** window for Surface, locate the **Expression** section.

2 In the **Expression** text field, type `1`.

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

4 From the **Color** list, choose **Black**.

3D Plot Group 1

Right-click **3D Plot Group 1** and choose **Arrow Surface**.

Arrow Surface 1

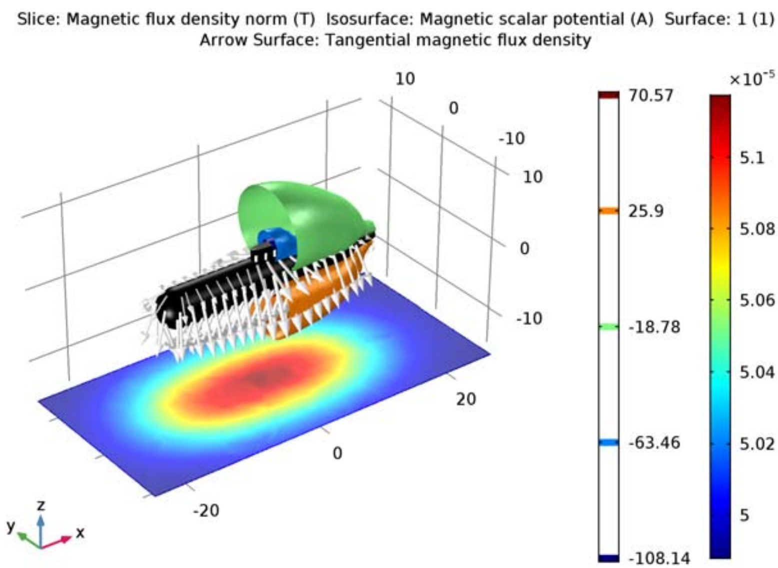
1 In the **Settings** window for Arrow Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Fields, No Currents>Magnetic>mfnc.tBx,...,mfnc.tBz - Tangential magnetic flux density**.

2 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

3 On the **3D Plot Group 1** toolbar, click **Plot**.

4 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

The plot shows the effect of the submarine's hull on the geomagnetic field.



Superconducting Wire

Introduction

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 application solves a time-dependent problem of a current building up in a superconducting wire close to the critical current density. This application is based on a suggestion by Dr. Roberto Brambilla, CESI, Superconductivity Dept., Milano, Italy.

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 application 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 Magnetic Fields interface. The reason is that 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

$$\mathbf{J} = \nabla \times \mathbf{H}$$

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

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

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

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

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 available in the AC/DC Module as the Magnetic Field Formulation interface.

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

$$
\oint \mathbf{H} \cdot \mathbf{dl} = 2\pi r H_{\phi} = I_{\text{wire}} \Rightarrow H_{\phi} = \frac{I_{\text{wire}}}{2\pi r}
$$

This is applied as a constraint on the tangential component of the vector field.

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.

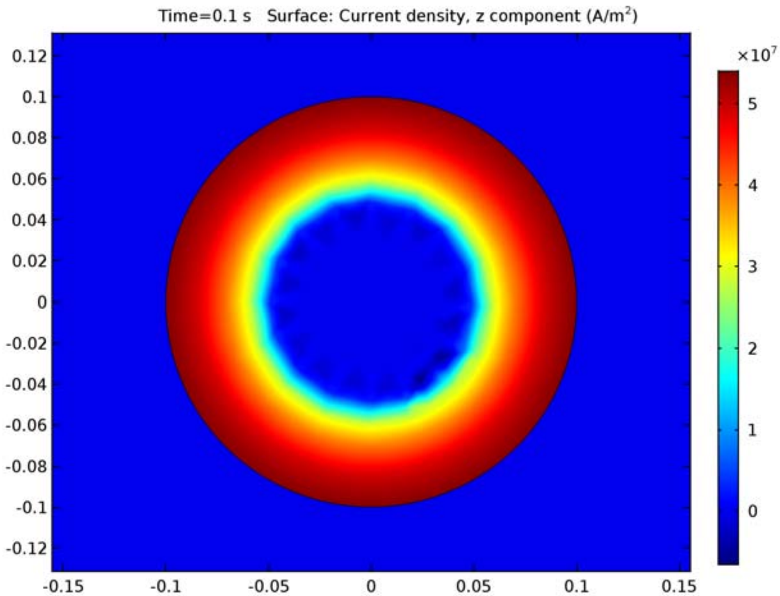


Figure 1: The current density at 0.1 s.

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.

Application Library path: `ACDC_Module/Other_Industrial_Applications/superconducting_wire`

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 **2D**.
2 In the **Select Physics** tree, select **AC/DC>Magnetic Field Formulation (mfh)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6 Click **Done**.

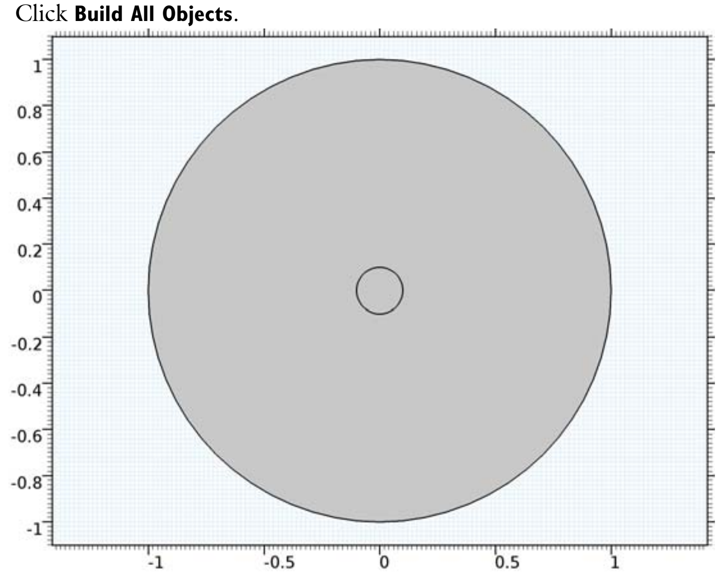
GEOMETRY 1

Circle 1 (c1)

On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
2 In the **Settings** window for Circle, locate the **Size and Shape** section.
3 In the **Radius** text field, type `0.1`.
4 Click **Build All Objects**.



GLOBAL DEFINITIONS

Parameters

1. On the **Home** toolbar, click **Parameters**.
2. In the **Settings** window for Parameters, locate the **Parameters** section.
3. In the table, enter the following settings:

Name	Expression	Value	Description
alpha	1.449621256	1.4496	Parameter for resistivity model
Jc	1.7e7[A/m^2]	1.7E7 A/m²	Critical current density
I0	1e6[A]	1E6 A	Applied current
rho_air	1e2[ohm*m]	100 Ω·m	Resistivity of air
tau	0.02[s]	0.02 s	Time constant for applied current
Tc	92[K]	92 K	Critical temperature
dT	4[K]	4 K	Parameter for resistivity model
dJ	Jc/1e4	1700 A/m²	Parameter for resistivity model
E0	0.0836168[V/m]	0.083617 V/m	Parameter for resistivity model

DEFINITIONS

Define a step function that will be used in the expression of the superconductor characteristic.

Step 1 (step1)

On the **Home** toolbar, click **Functions** and choose **Local>Step**.

Variables 1

1. On the **Home** toolbar, click **Variables** and choose **Local Variables**.
2. In the **Settings** window for Variables, locate the **Variables** section.
3. In the table, enter the following settings:

Name	Expression	Unit	Description
I1	I0*(1-exp(-t/tau))	A	
H0phi	I1/(2*pi*sqrt(x^2+y^2))	A/m	

COMPONENT 1 (COMP1)

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

DEFINITIONS

In order to simplify the application of the boundary condition, add a cylindrical coordinate system.

Cylindrical System 2 (sys2)

On the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.

MAGNETIC FIELD FORMULATION (MFH)

When using the **Magnetic Field Formulation** physics with superconductive material, it is necessary to turn off the automatic divergence constraint, that can lead to instability.

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.
2 In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Field Formulation (mfh)**.
3 In the **Settings** window for Magnetic Field Formulation, click to expand the **Divergence constraint** section.
4 Locate the **Divergence Constraint** section. Clear the **Activate divergence constraint** check box.

Faraday's Law 1

Set the constitutive relation for the default Faraday's Law node to use **Resistivity**.

1 In the **Model Builder** window, under **Component 1 (comp1)>Magnetic Field Formulation (mfh)** click **Faraday's Law 1**.
2 In the **Settings** window for Faraday's Law, locate the **Conduction Current** section.
3 From the **Conduction current relation** list, choose **Resistivity**.

Faraday's Law 2

1 On the **Physics** toolbar, click **Domains** and choose **Faraday's Law**.
2 Select Domain 2 only.
3 In the **Settings** window for Faraday's Law, locate the **Conduction Current** section.
4 From the **Conduction current relation** list, choose **E-J characteristic**.

   The Faraday's Law feature will use the material data specified in the Superconductor material.

   Set up the boundary condition for the magnetic field.

Magnetic Field 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Magnetic Field**.

2 Select Boundaries 1, 2, 5, and 8 only.

3 In the **Settings** window for Magnetic Field, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

5 Locate the **Magnetic Field** section. Specify the $\mathbf{H}_0$ vector as

`0`	r
`H0phi`	phi
`0`	a

MATERIALS

Add the materials used in the model. For the domain surrounding the wire, create a material representing Air.

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 Right-click **Material 1 (mat1)** and choose **Rename**.

3 In the **Rename Material** dialog box, type `Air` in the **New label** text field.

4 Click **OK**.

The **Magnetic Field Formulation** physics requires a finite resistivity in all domains.

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

6 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	1	1	Basic
Relative permeability	mur	1	1	Basic
Resistivity	res	`rho_air`	Ω·m	Basic

For the superconductor, create a custom material that uses the 'E-J Characteristic' model.

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 Right-click **Material 2 (mat2)** and choose **Rename**.

3 In the **Rename Material** dialog box, type `Superconductor` in the **New label** text field.

4 Click **OK**.

5 Select Domain 2 only.

Fill in the relative permittivity and permeability.

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

7 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Relative permittivity	epsilonr	1	1	Basic
Relative permeability	mur	1	1	Basic

Now add a subnode that provides the material model for the superconductor.

8 Click to expand the **Material properties** section. Locate the **Material Properties** section. In the **Material properties** tree, select **Electromagnetic Models>E-J Characteristic**.

9 Click **Add to Material**.

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

Property	Name	Value	Unit	Property group
Electric field norm	normE	`E0*(((normJ-Jc)/Jc)*step1((normJ-Jc)/1[A/m^2]))^alpha`	V/m	E-J characteristic

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

Free Triangular 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

Size 1

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

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

3 Select Domain 2 only.

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

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `0.02`.

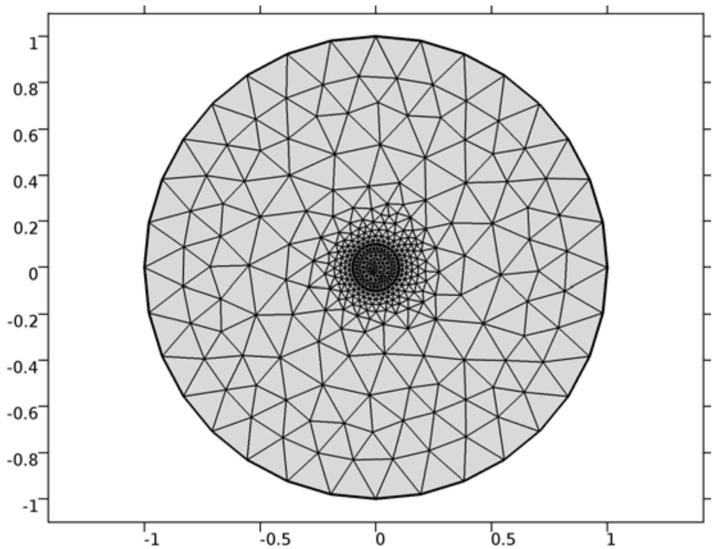
Size

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

2 In the **Settings** window for Size, locate the **Element Size** section.

3 From the **Predefined** list, choose **Coarse**.

4 Click **Build All**.



STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

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

3 Click **Range**.

4 In the **Range** dialog box, type `0.005` in the **Step** text field.

5 In the **Stop** text field, type `0.1`.

6 Click **Replace**.

7 In the **Settings** window for Time Dependent, locate the **Study Settings** section.

8 Select the **Relative tolerance** check box.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

4 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

5 Locate the **Time Stepping** section. Select the **Initial step** check box.

6 In the associated text field, type `1e-9`.

7 Select the **Maximum step** check box.

8 In the associated text field, type `1e-3`.

9 On the **Study** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density Norm (mfh)

When the computation is finished, the default plot shows the norm of the magnetic flux density.

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

2D Plot Group 2

On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

Surface 1

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

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Magnetic Field Formulation>Currents and charge>Current density>mfh.Jz - Current density, z component**.

3 On the **2D Plot Group 2** toolbar, click **Plot**.

4 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.

Animation 1

1 On the **Results** toolbar, click **Animation** and choose **File**.

2 In the **Settings** window for Animation, locate the **Target** section.

3 From the **Target** list, choose **Player**.

4 Locate the **Scene** section. From the **Subject** list, choose **2D Plot Group 2**.

5 Right-click **Animation 1** and choose **Play**.