Created in COMSOL Multiphysics 5.6

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.

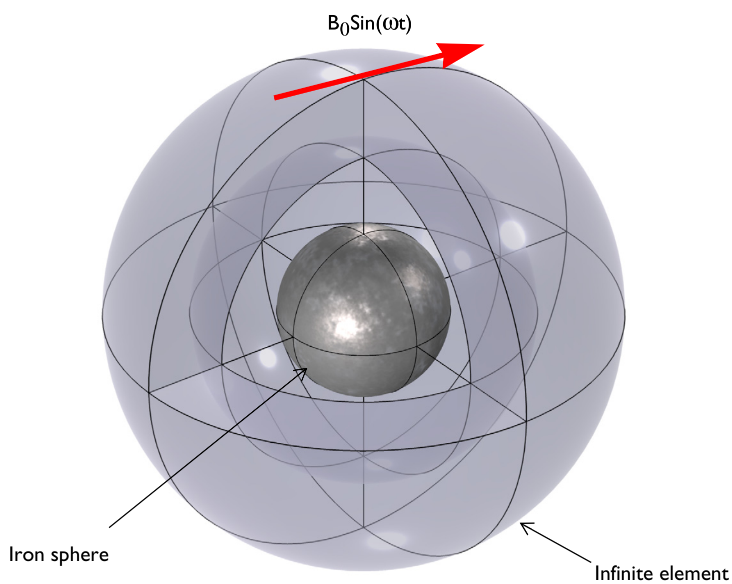


Figure 1: An iron sphere is exposed to a spatially uniform, sinusoidally time-varying, background magnetic field.

Model Definition

Figure 1 shows the setup, with an iron sphere placed in a spatially uniform time-harmonic background magnetic field. The background field is applied using the Reduced field formulation available in the Magnetic Fields interface. The model space is truncated by an Infinite Elements region, a domain condition approximating a domain that extends to infinity. When using Infinite Element Domain features, the boundary condition on the outside of the modeling domain only marginally affects the solution, since it is placed at a large physical distance.

The iron sphere has a relative permittivity of $\varepsilon_r = 1$, a relative permeability of $\mu_r = 4000$, and an electric conductivity of $\sigma = 1.12 \cdot 10^7$ 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.

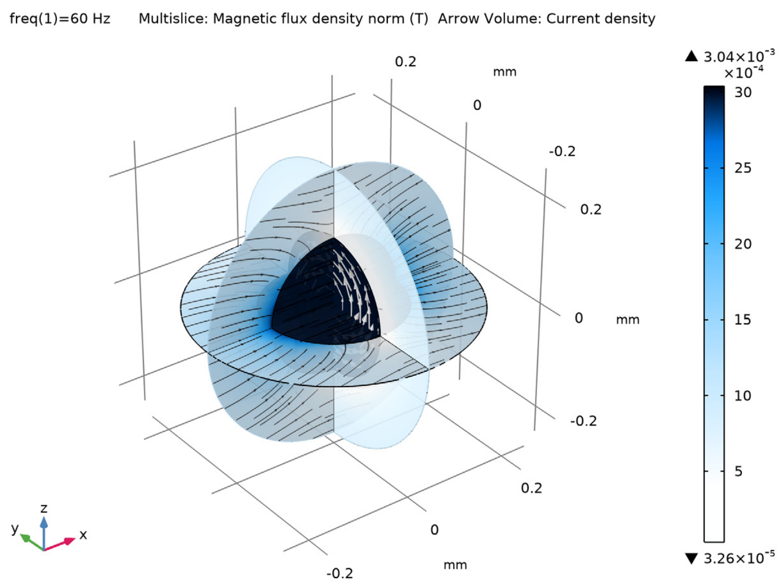


Figure 2: The induced currents and the magnetic field for the model without gauge fixing.

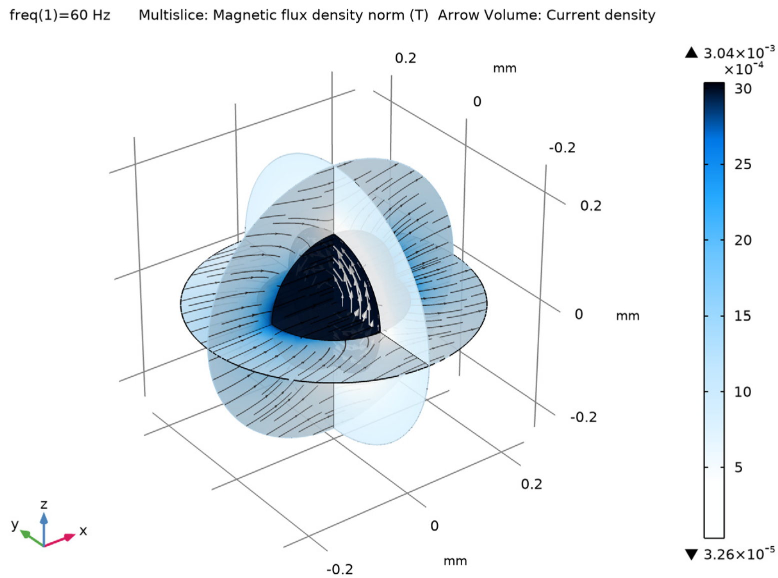


Figure 3: The induced currents and the magnetic field for the model with gauge fixing.

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>Electromagnetic Fields>Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
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**.

Sphere 1 (sph1)

Create a sphere with two layers plus an inner core. The outermost layer represents the exterior air region, scaled using the Infinite Element Domain, the middle layer is the unscaled air domain, and the core represents the iron sphere.

1 In the **Geometry** toolbar, click **Sphere**.

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

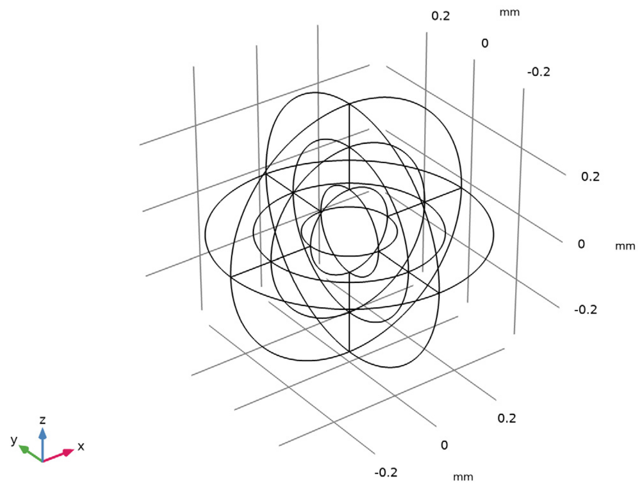
3 In the **Radius** text field, type 3*r0.

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 in the **Graphics** toolbar.



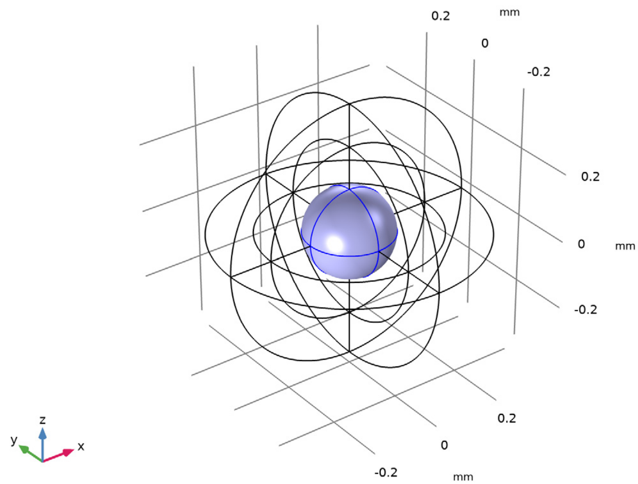
DEFINITIONS

Create a set of selections before setting up the physics. First, create a selection for the surface of the iron sphere.

Core

1 In 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.

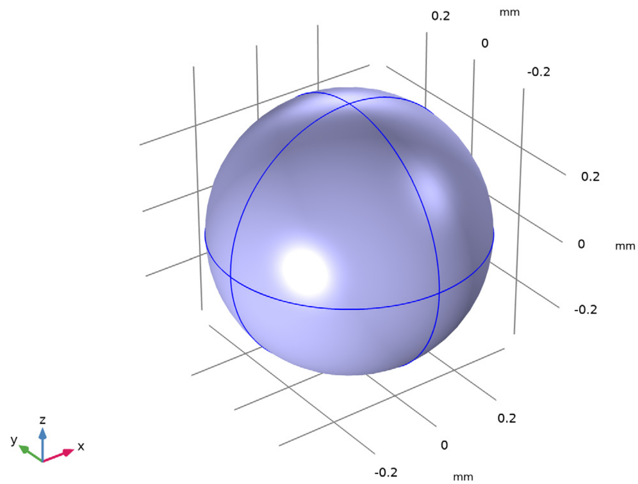
5 Click **OK**.

Add a selection for the Infinite Element Domain feature.

Infinite Element domains

1 In 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.

Analysis domain

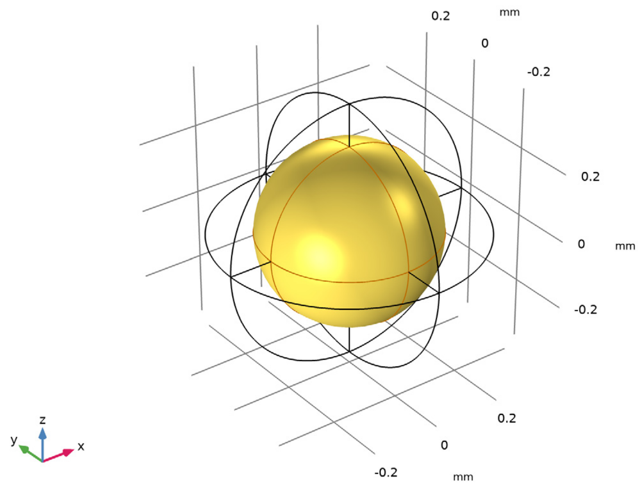
1 In the **Definitions** toolbar, click **Complement**.

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 node. Use the selection **Infinite Element domains**.

Infinite Element Domain 1 (ie1)

1 In 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.

Air

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

In order to improve the convergence rate of the solver, 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	Variable	Value	Unit	Property group
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	5000	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

4 Right-click **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 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in>Iron**.

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

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

MATERIALS

Iron (mat2)

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

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

MESH 1

The **Magnetic Fields** interface's **Physics induced mesh** creates a swept mesh for the Infinite Element Domain.

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

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

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

4 Click **Build All**.

Plot the meshed structure to review the quality of the mesh.

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

RESULTS

Mesh 1

By default, the boundary mesh is plotted, so only the triangular elements on the outer boundaries are visible. Perform the following operations to inspect the tetrahedral elements in the interior of the geometry.

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

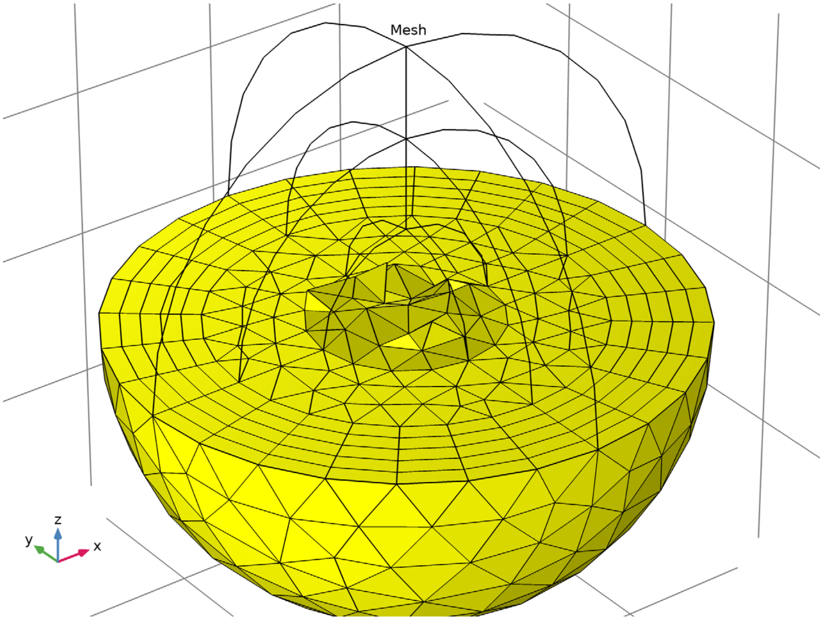
2 From the **Element color** list, choose **Yellow**.

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

4 In the **Expression** text field, type `z<0` to plot a section of the mesh.

5 In the **Mesh Plot 1** toolbar, click **Plot**.

6 Click the **Zoom In** button in 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 In 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>Datasets** node, then click **Study 1/ Solution 1 (sol1)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **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**.

Arrow Volume 1

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

2 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 (comp1)>Magnetic Fields>Currents and charge>mf.Jx,mf.Jy,mf.Jz - Current density**.

3 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `1`.

4 Find the **y grid points** subsection. In the **Points** text field, type `21`.

5 Find the **z grid points** subsection. In the **Points** text field, type `21`.

6 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.

Skin depth (mf)

1 In 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

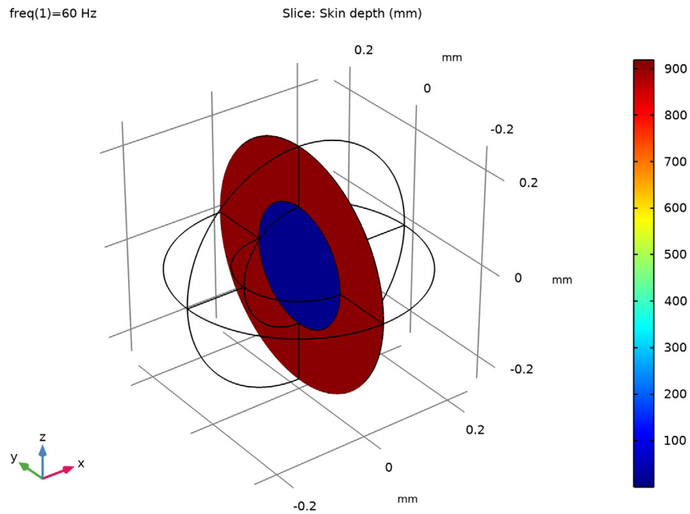
1 Right-click **Skin depth (mf)** and choose **Slice**.

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 (comp1)>Magnetic Fields>Material properties>mf.deltaS - Skin depth - m**.

5 In 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	**Variable**	**Value**	**Unit**	**Property group**
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	5	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

ADD STUDY

1 In 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 **General Studies> Frequency Domain**.

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

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

MAGNETIC FIELDS (MF)

Gauge Fixing for A-field 1

In the **Physics** toolbar, click **Domains** and choose **Gauge Fixing for A-field**.

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 solver setting adjustment described below is optional.

Solution 2 (sol2)

1 In 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 by direct linear solver MUMPS to determine 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. Specifying a larger allocation factor before solving will prevent the solver from having to perform multiple tries, potentially saving some time. For more information, see the documentation for the MUMPS direct solver.

6 In the **Study** toolbar, click **Compute**.

RESULTS

Specify a selection for the second solution and add an arrow and a skin depth plot as it was done for first plot.

Study 2/Solution 2 (sol2)

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

Selection

1 In the **Results** toolbar, click **Attributes** and choose **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**.

Arrow Volume 1

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

2 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 (comp1)> Magnetic Fields>Currents and charge>mf.Jx,mf.Jy,mf.Jz - Current density**.

3 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type 1.

4 Find the **y grid points** subsection. In the **Points** text field, type 21.

5 Find the **z grid points** subsection. In the **Points** text field, type 21.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Gray**.

7 In the **Magnetic Flux Density Norm (mf) 1** toolbar, click **Plot**.

Compare the reproduced plot with Figure 3.

Skin depth (mf) 1

1 In 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 **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

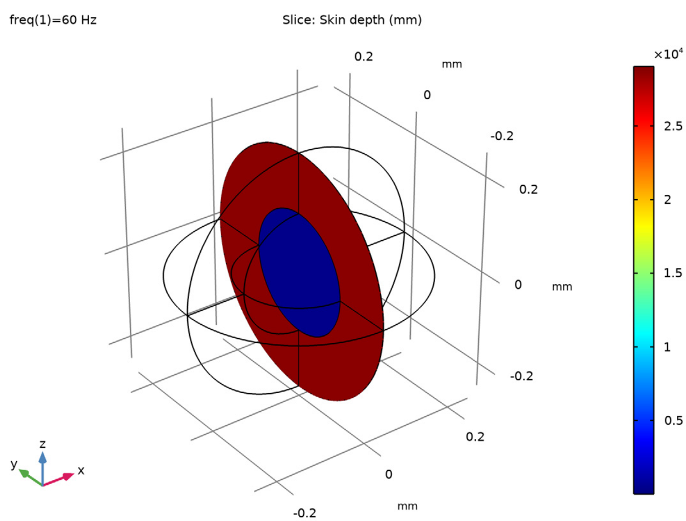
Slice 1

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 (comp1)>Magnetic Fields> Material properties>mf.deltaS - Skin depth - m**.

3 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

4 In the **Skin depth (mf) 1** toolbar, click **Plot**.



Finally, evaluate the total dissipation by integrating the resistive losses on each air and iron domain.

Volume Integration 1

1 In 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 (comp1)>Magnetic Fields>Heating and losses>mf.Qrh - Volumetric loss density, electric - W/m³**.

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 In 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 **Dataset** 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 (comp1)>Magnetic Fields>Heating and losses>mf.Qrh - Volumetric loss density, electric - W/m³**.

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

Expression	Unit	Description
mf.Qrh	nW	Volumetric loss density, electric

7 Click **Evaluate (Table 1 - Volume Integration 1)**.

Volume Integration 3

1 In 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 (comp1)>Magnetic Fields>Heating and losses>mf.Qrh - Volumetric loss density, electric - W/m³**.

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

Expression	Unit	Description
mf.Qrh	nW	Volumetric loss density, electric

6 Click **Evaluate (Table 1 - Volume Integration 1)**.

Volume Integration 4

1 In 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 **Dataset** 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 (comp1)>Magnetic Fields>Heating and losses>mf.Qrh - Volumetric loss density, electric - W/m³**.

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

Expression	Unit	Description
mf.Qrh	nW	Volumetric loss density, electric

7 Click **Evaluate (Table 1 - Volume Integration 1)**.