Created in COMSOL Multiphysics 5.6

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.

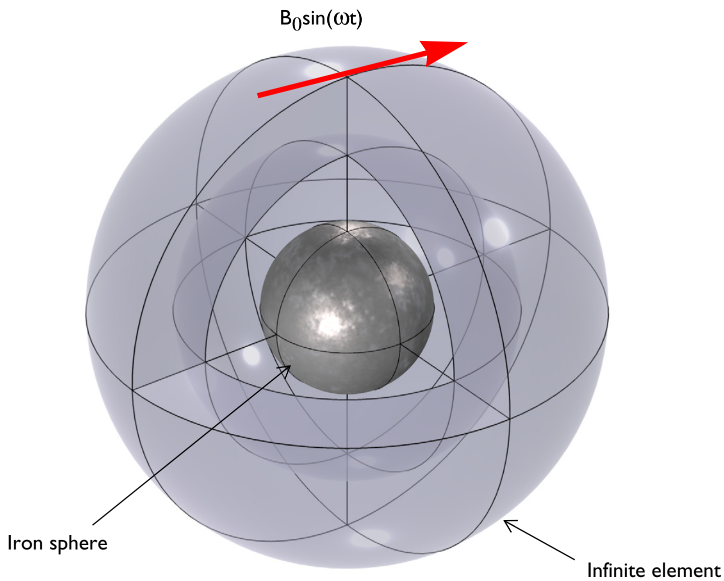


Figure 1: An iron sphere is exposed to a spatially uniform, sinusoidally time-varying, background magnetic field.

Model Definition

Figure 1 shows the setup, with an iron sphere placed in a spatially uniform 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 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, an operation which creates short triangular prismatic elements along the direction normal to the surface. 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.

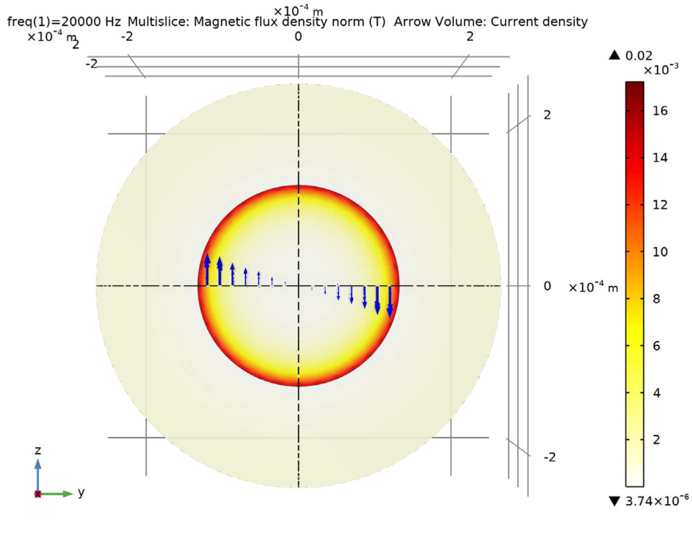


Figure 2: The induced currents and the magnetic field in the iron sphere.

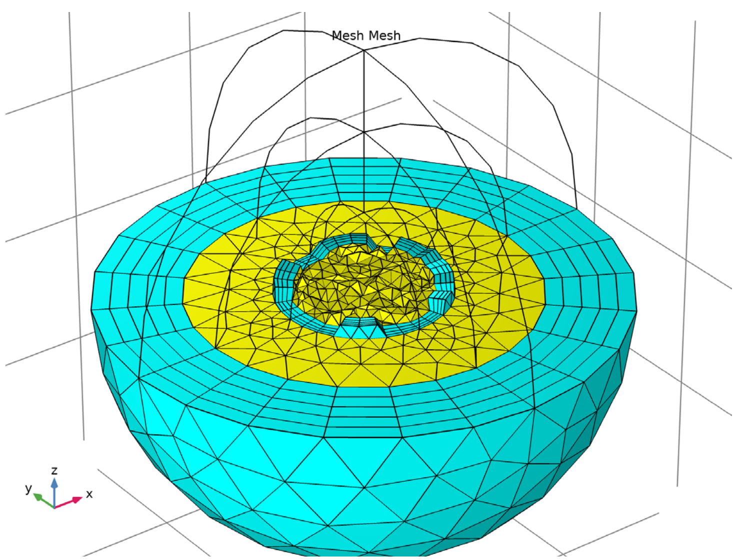


Figure 3: The mesh in the iron sphere and surrounding air.

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>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

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.

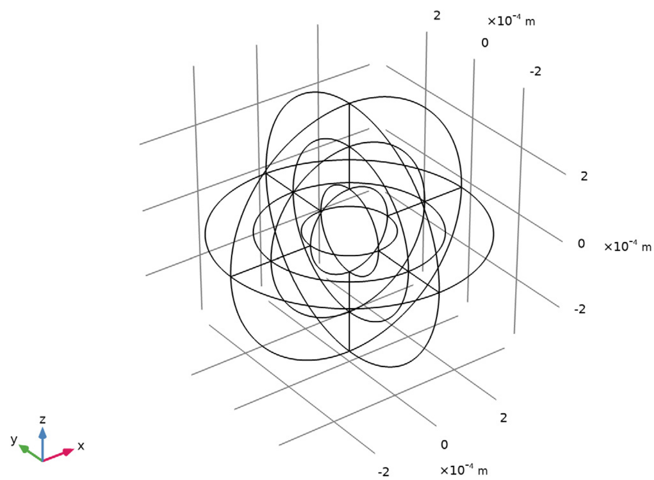
3 In the **Radius** text field, type `3*r0`.

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

Layer name	Thickness (m)
Layer 1	r0
Layer 2	r0

5 Click **Build All Objects**.

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



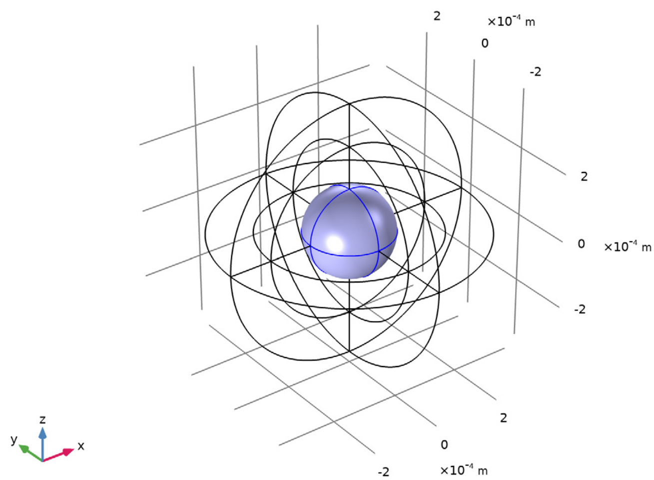
DEFINITIONS

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.

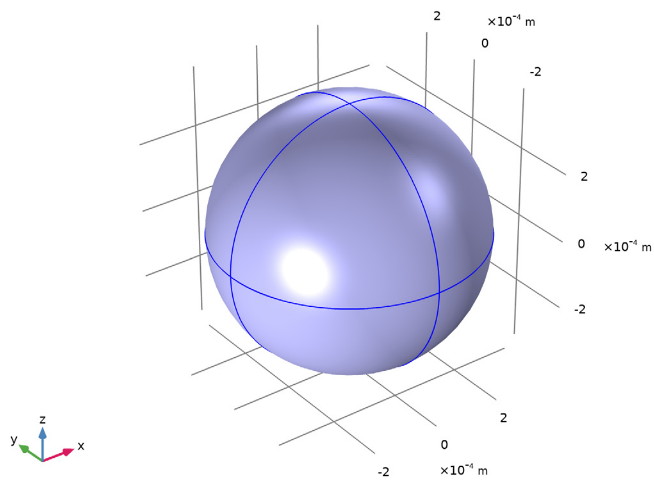
5 Click **OK**.

Add a selection for the Infinite Element Domain feature.

Infinite Element domain

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 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 domain** selection.

Analysis domain

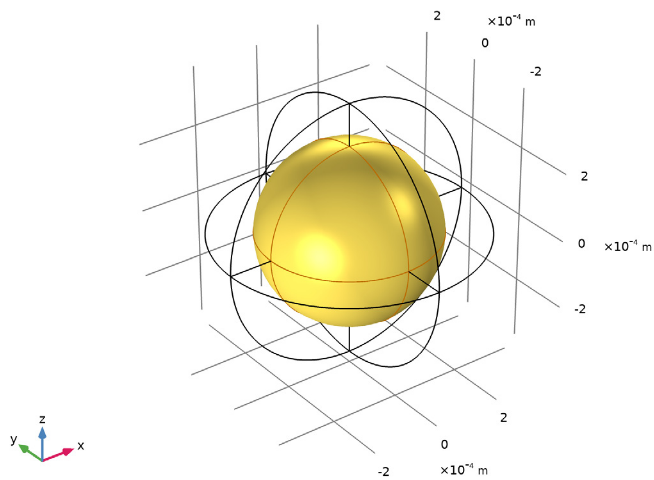
1 In the **Definitions** toolbar, click **Complement**.

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 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 domain**.

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 domain**.

4 Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

MAGNETIC FIELDS (MF)

Set up the physics interface to apply a uniform background magnetic fields. In the **Magnetic Fields** interface, the background field must be specified in terms of a 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

Assign the 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**.

Specify the conductivity of the air to a small value in order to improve the convergence rate.

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	50	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

Size

Specify an extra fine mesh on the surface of the iron sphere.

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size 1

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

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

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

4 Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.

Boundary Layers 1

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

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

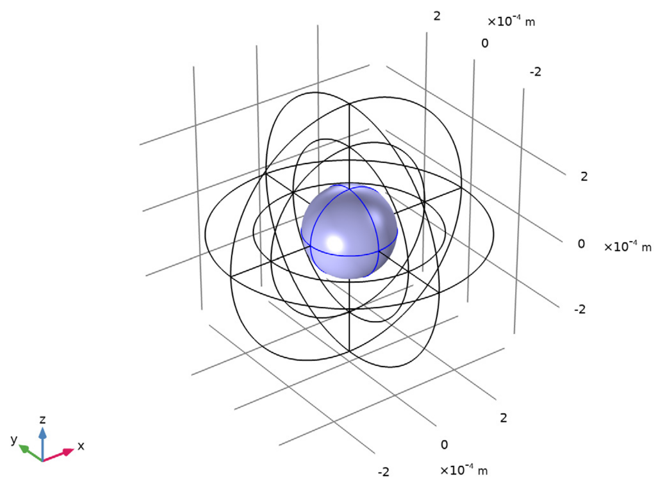
3 From the **Geometric entity level** list, choose **Domain**.

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

Boundary Layer Properties

1 In the **Model Builder** window, 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 From the **Thickness of first layer** list, choose **Manual**.

5 In the **Thickness** text field, type `8[um]`.

6 In the **Number of boundary layers** text field, type `4`.

7 Click **Build All**.

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

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

RESULTS

Mesh 1

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

2 In the **Settings** window for **3D Plot Group**, type `Mesh 1` in the **Label** text field.

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

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

3 From the **Element type** list, choose **Prism**.

4 Locate the **Coloring and Style** section. From the **Element color** list, choose **Cyan**.

5 Click to expand 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.

Mesh 2

1 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 **Coloring and Style** 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 In the **Mesh 1** toolbar, click **Plot**.

9 Click the **Zoom In** button in 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 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.

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**.

Streamline Surface 1, Streamline Surface 2, Streamline Surface 3

1 In the **Model Builder** window, under **Results>Magnetic Flux Density Norm (mf)**, Ctrl-click to select **Streamline Surface 1**, **Streamline Surface 2**, and **Streamline Surface 3**.

2 Right-click and choose **Disable**.

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 In the **Arrow Positioning** section, specify the grid points according to the following table:

Direction	Value
x	31
y	31
z	1

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

5 Click the **Go to YZ View** button in the **Graphics** toolbar.

Compare the reproduced plot with Figure 3.

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

7 Find the **z grid points** subsection. In the **Points** text field, type 31.

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