Created in COMSOL Multiphysics 6.1

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.

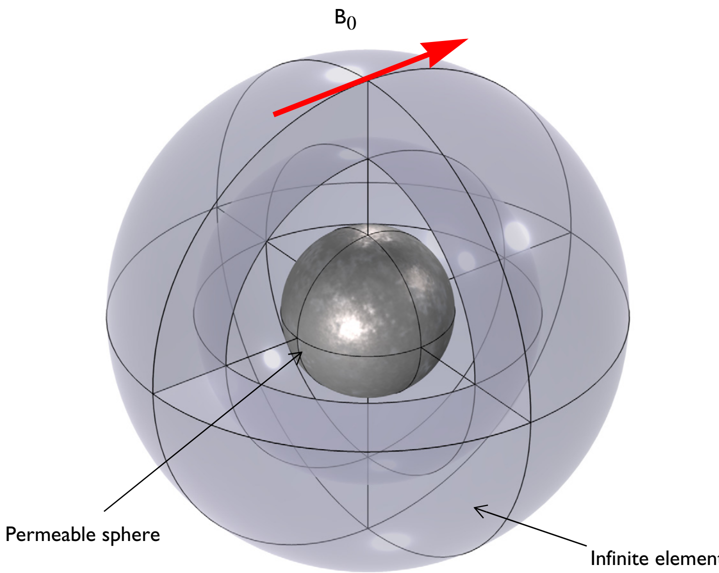


Figure 1: A magnetically permeable sphere in a spatially uniform, static background magnetic field. The sphere at the center is surrounded by air, and enclosed in a region of Infinite Elements.

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 μ_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 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.

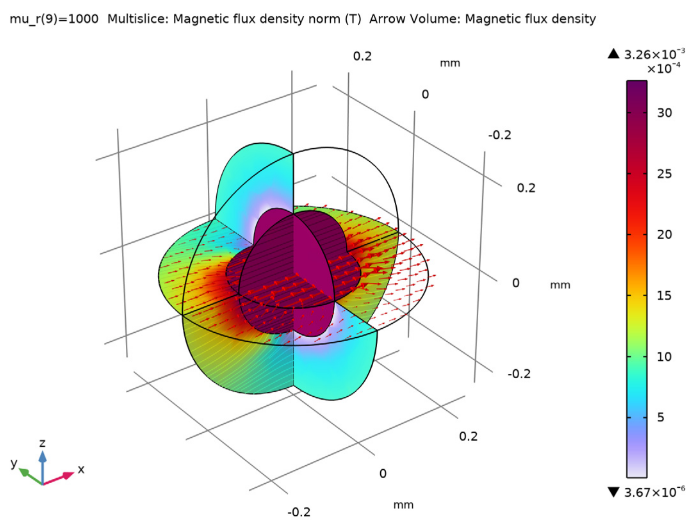


Figure 2: The magnetic field for the Magnetic Fields, No Currents interface.

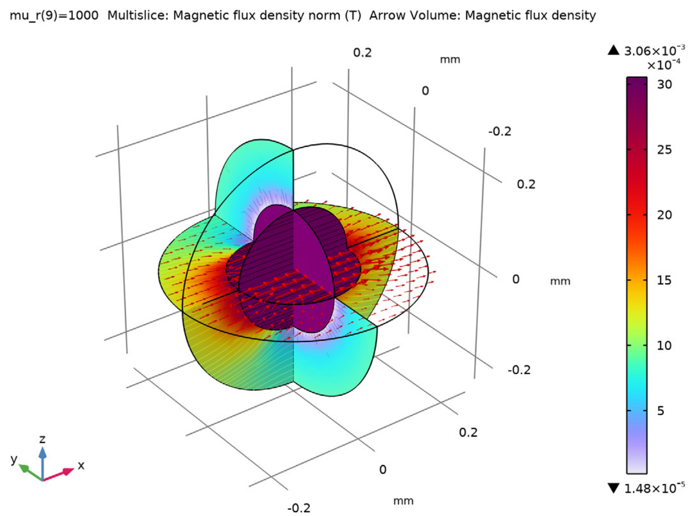


Figure 3: The magnetic field for the Magnetic Fields interface.

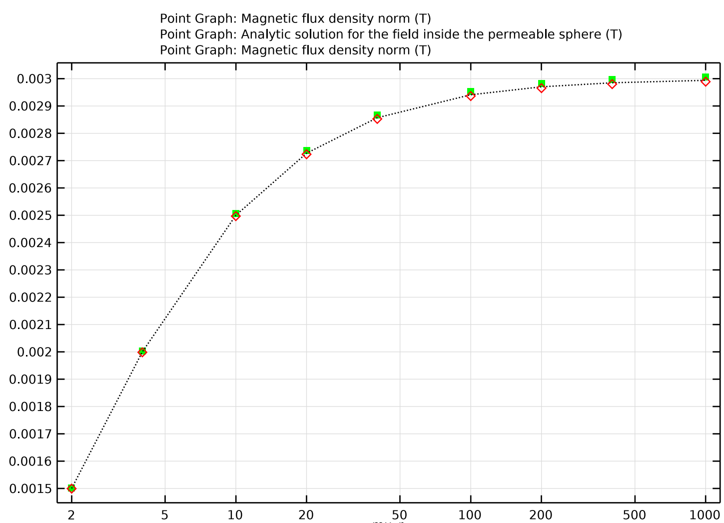


Figure 4: Comparison of numerical results to analytic result.

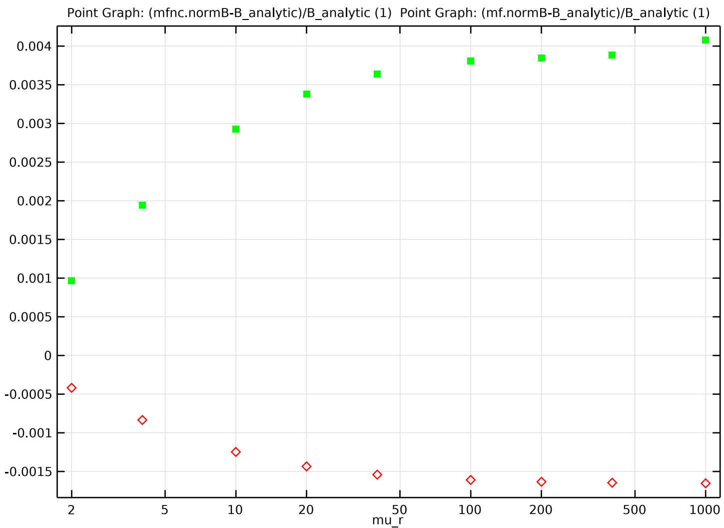


Figure 5: Relative difference compared to the analytic solution.

Application Library path: `ACDC_Module/Introductory_Magnetostatics/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>Magnetic Fields, No Currents (mfnc)**.
3. Click **Add**.
4. Click **Study**.

5 In the **Select Study** tree, select **General 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

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
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

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 median layer is the unscaled air domain, and the core represents the permeable 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 `r0*3`.

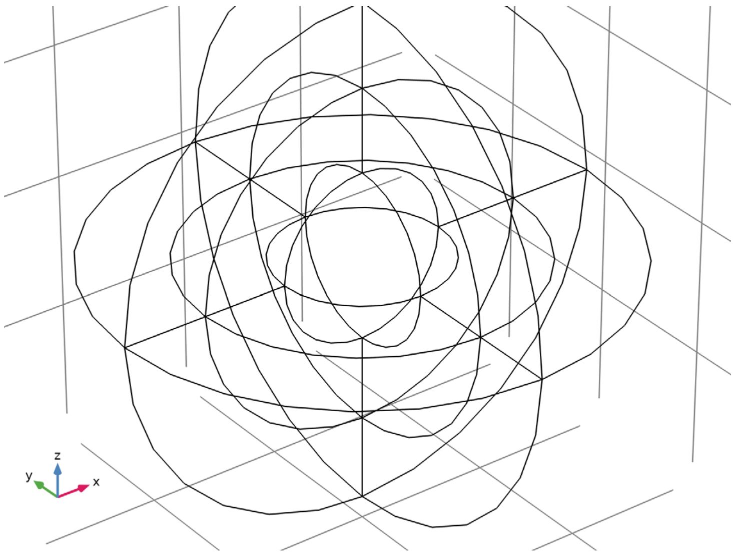
4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	`r0`
`Layer 2`	`r0`

5 Click **Build All Objects**.

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

7 Click the **Zoom In** button in the **Graphics** toolbar.



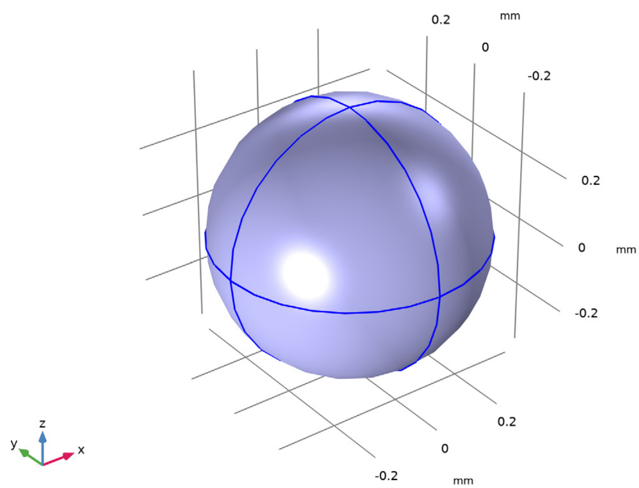
DEFINITIONS

Create a set of selections before setting up the physics. First, create 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 In the **Settings** window for **Explicit**, type `Infinite Element domains` in the **Label** text field.

Analysis domain

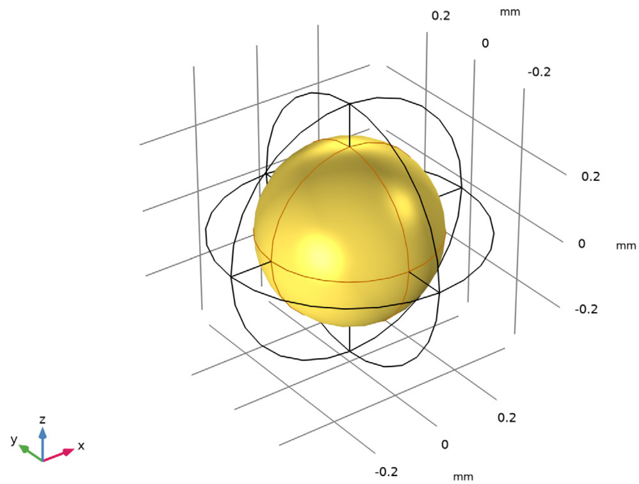
1 In the **Definitions** toolbar, click **Complement**.

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 In the **Settings** window for **Complement**, type `Analysis domain` in the **Label** text field.

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**.

DEFINITIONS

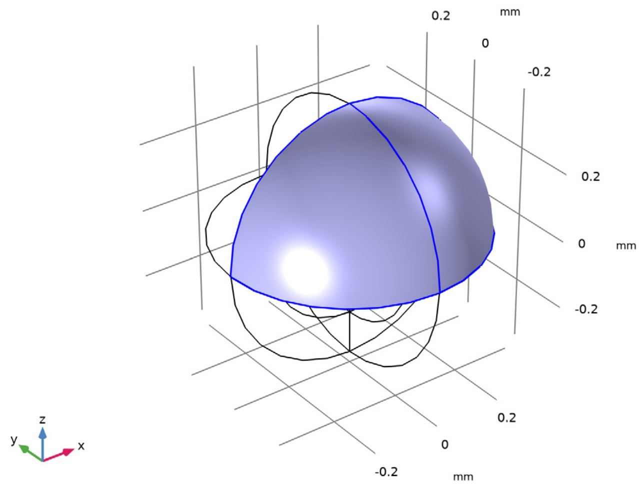
View 1

Suppress some domains to get a better view when setting up the physics and reviewing the meshed results.

Hide for Physics 1

1 In the **Model Builder** window, right-click **View 1** and choose **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

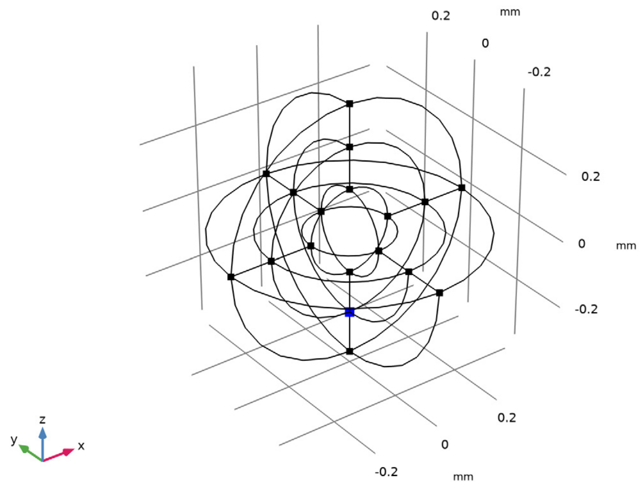
B0/mu0_const	x
0	y
0	z

Add a constraint point for the magnetic scalar potential.

Zero Magnetic Scalar Potential 1

1 In the **Physics** toolbar, click **Points** and choose **Zero Magnetic Scalar Potential**.

2 Select Point 8 only.



MATERIALS

Assign the material properties. First, use air for all domains.

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>Air**.

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

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

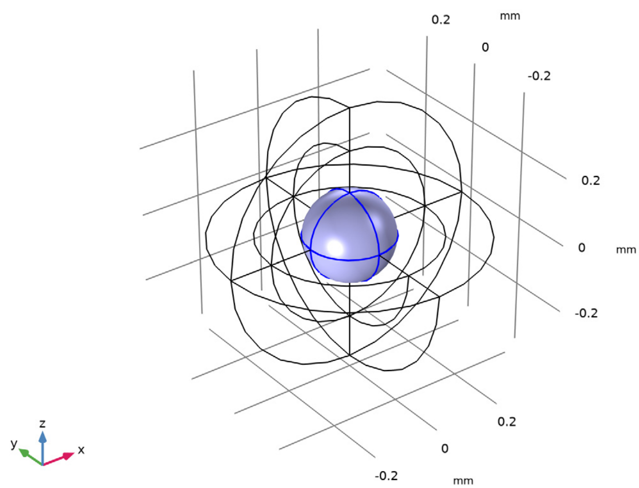
MATERIALS

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

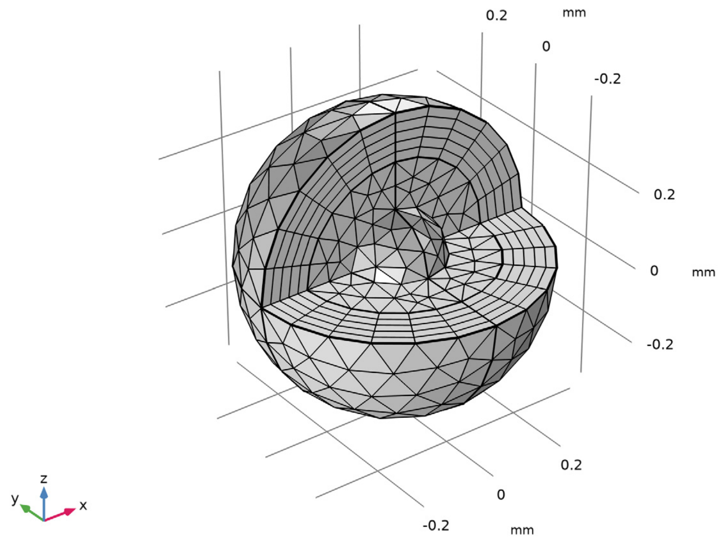
4 In the table, enter the following settings:

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

The **Physics-controlled mesh** setting creates a swept mesh for the Infinite Element Domain.

MESH 1

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



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**, click to expand the **Study Extensions** section.

3 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 (Relative permeability, magnetically permeable sphere)	2 4 10 20 40 100 200 400 1000	

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

RESULTS

Study 1/Solution 1 (sol1)

The default plot shows the magnetic flux density for all domains. Add a selection to the current solution to visualize only the Analysis domain.

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**.

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

   Compare the plot with Figure 2.

Repeat the analysis with the **Magnetic Fields** interface.

ADD PHYSICS

1 In 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>Electromagnetic Fields>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 1** in the window toolbar.
6 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

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>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Magnetic Fields, No Currents (mfnc)**.

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

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

STUDY 2

Step 1: Stationary

Set up the Magnetic Fields interface. Specify the background field in terms of a magnetic vector potential.

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 to the permeable 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	**Variable**	**Value**	**Unit**	**Property group**
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	`mu_r`	1	Basic

Add manually a different mesh for the second physics interface starting from the physics induced sequence.

MESH 2

In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

Size

Right-click **Mesh 2** and choose **Edit Physics-Induced Sequence**.

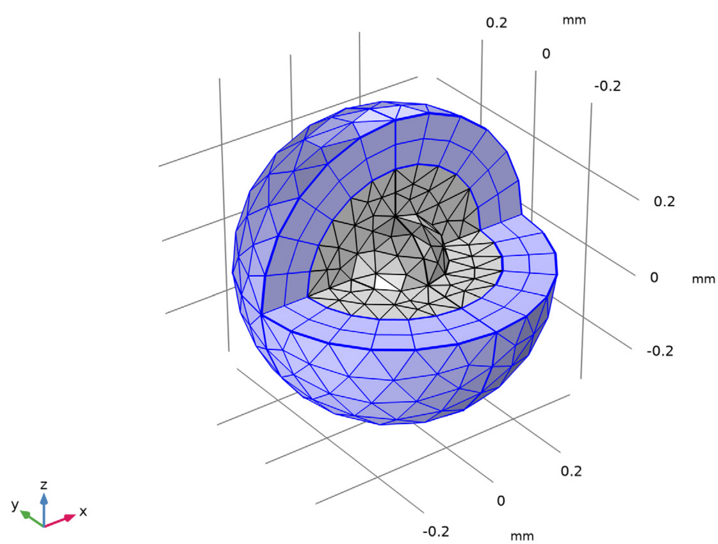
Distribution 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Meshes>Mesh 2>Swept 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 2.

4 Click **Build All**.



STUDY 2

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 **Mesh Selection** section.

3 In the table, enter the following settings:

Component	Mesh
Component 1	Mesh 2

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 (Relative permeability, magnetically permeable sphere)	2 4 10 20 40 100 200 400 1000	

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

RESULTS

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**.

Add a Cut Point 3D dataset for both physics. At the origin, the numerical and analytical results are evaluated.

Cut Point 3D 1

1 In 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 In the **Results** toolbar, click **Cut Point 3D**.

2 In the **Settings** window for **Cut Point 3D**, locate the **Data** section.

3 From the **Dataset** 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.

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>Magnetic>mf.Bx,mf.By,mf.Bz - Magnetic flux density**.

3 Locate the **Arrow Positioning** section. 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 In the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

The plot should look like Figure 3.

1D Plot Group 4

1 In 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 Right-click **1D Plot Group 4** and choose **Point Graph**.
2 In the **Settings** window for **Point Graph**, locate the **Data** section.
3 From the **Dataset** 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. 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**.

Point Graph 2

1 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Point Graph**.
2 In the **Settings** window for **Point Graph**, locate the **Data** section.
3 From the **Dataset** 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**.

Point Graph 3

1 Right-click **1D Plot Group 4** and choose **Point Graph**.
2 In the **Settings** window for **Point Graph**, locate the **Data** section.
3 From the **Dataset** 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 In the **1D Plot Group 4** toolbar, click **Plot**.

1D Plot Group 5

1 In 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 Right-click **1D Plot Group 5** and choose **Point Graph**.

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

3 From the **Dataset** 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**.

Point Graph 2

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

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

3 From the **Dataset** 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 In the **1D Plot Group 5** toolbar, click **Plot**.

Compare the resulting plots with Figure 4 and Figure 5.