Created in COMSOL Multiphysics 5.6

Magnetic Field of a Helmholtz Coil

Introduction

A Helmholtz coil is a parallel pair of identical circular coils spaced one radius apart and wound so that the current flows through both coils in the same direction. This winding results in a uniform magnetic field between the coils with the primary component parallel to the axis of the two coils. The uniform field is the result of the sum of the two field components parallel to the axis of the coils and the difference between the components perpendicular to the same axis.

The purpose of the device is to allow scientists and engineers to perform experiments and tests that require a known ambient magnetic field. Helmholtz field generation can be static, time-varying DC, or AC, depending on application.

Applications include canceling the Earth's magnetic field for certain experiments; generating magnetic fields for determining magnetic shielding effectiveness or susceptibility of electronic equipment to magnetic fields; calibration of magnetometers and navigational equipment; and biomagnetic studies.

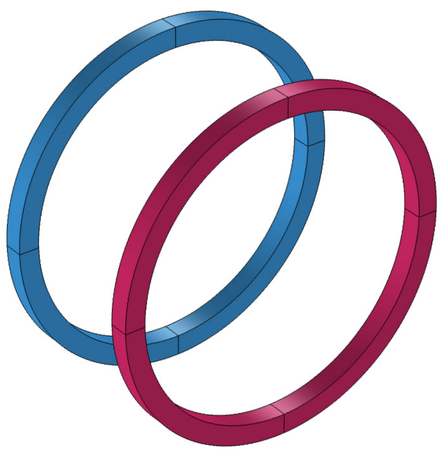


Figure 1: The Helmholtz coil consists of two coaxial circular coils, one radius apart along the axial direction. The coils carry parallel currents of equal magnitude.

Model Definition

The application shows how to compute the magnetic field with two different approaches, one using the **Magnetic Fields** interface and the other the **Magnetic Fields, Currents Only** interface. The model geometry is shown in Figure 2.

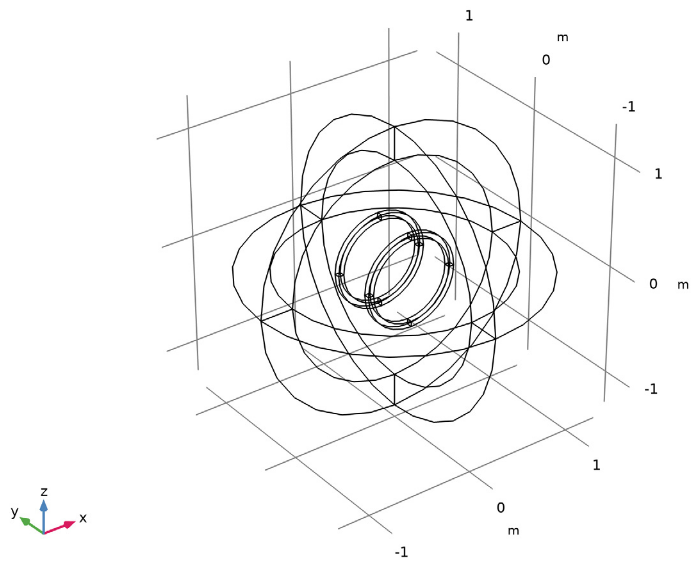


Figure 2: The model geometry.

DOMAIN EQUATIONS

Assuming static currents and fields, the magnetic vector potential $\mathbf{A}$ must satisfy the following equation:

$$\nabla \times (\mu^{-1} \nabla \times \mathbf{A}) = \mathbf{J}_e$$

where μ is the permeability, and $\mathbf{J}_e$ denotes the externally applied current density.

The relations between the magnetic field $\mathbf{H}$, the magnetic flux density $\mathbf{B}$ and the potential are given by

$$\mathbf{B} = \nabla \times \mathbf{A}$$

$$\mathbf{H} = \mu^{-1} \mathbf{B}$$

This model uses the permeability of vacuum, that is, $\mu \approx 4\pi\times10^{-7}$ H/m. The external current density is computed using a homogenized model for the coils, each one made by 10 wire turns and excited by a current of 0.25 mA. The currents are specified to be parallel for the two coils.

SPATIAL DERIVATIVE OF MAGNETIC FIELD

Computing the spatial derivative of the magnetic field or magnetic flux density is useful in areas such as radiology, magnetophoresis, particle accelerators, and geophysics. One of the most important use cases is the design of magnetic resonance imaging (MRI) machines, where it is necessary to analyze not only the field strength but also the spatial variation of the field. This application demonstrates how to compute the spatial derivative of the magnetic flux density in the post-processing step.

Results and Discussion

Figure 3 shows the magnetic flux density between the coils. The flux is relatively uniform in the region between the coils. This uniformity is the main property and often the sought feature of a Helmholtz coil.

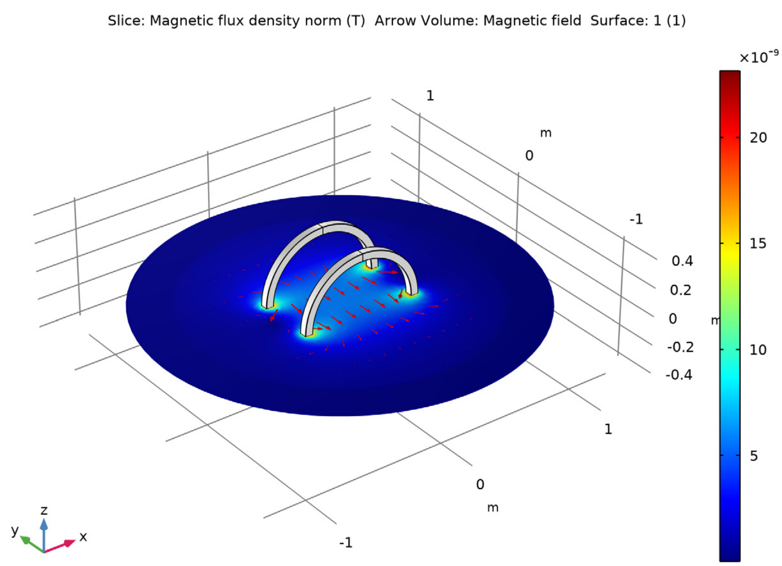


Figure 3: The slice plot shows the magnetic flux density. The arrows indicate the magnetic field (H) strength and direction.

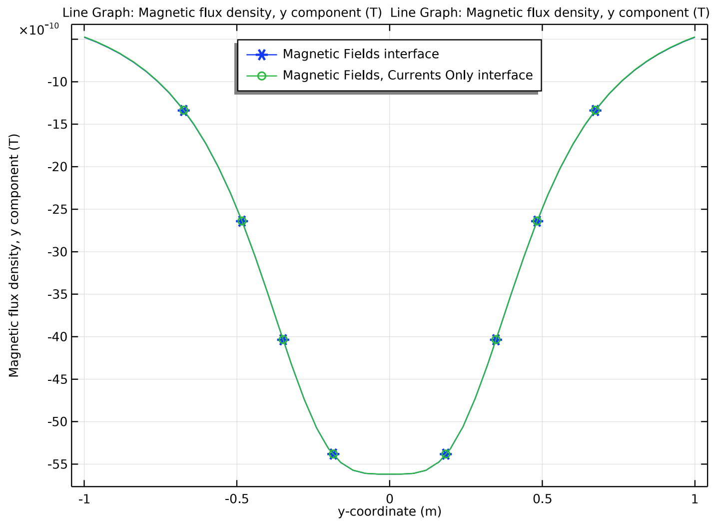


Figure 4: Comparison of the y component of the B field along the centerline of the Helmholtz coil using two different approaches.

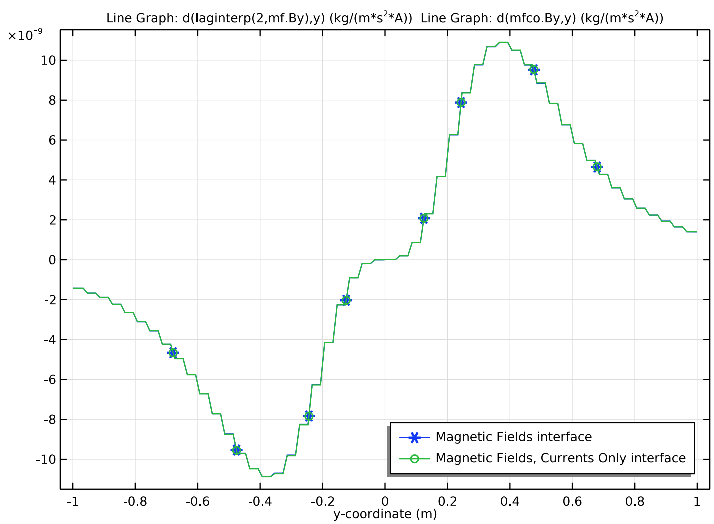


Figure 5: Comparison of the gradient (with respect to the y direction) of the y component of the B field along the centerline of the Helmholtz coil.

Application Library path: `ACDC_Module/Inductive_Devices_and_Coils/helmholtz_coil`

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 In the **Select Physics** tree, select **AC/DC>Electromagnetic Fields>Vector Formulations>Magnetic Fields, Currents Only (mfco)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Stationary**.

8 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
`I0`	`0.25[mA]`	2.5E-4 A	`Coil current`

GEOMETRY 1

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Square 1 (sq1)

1 In the **Work Plane** toolbar, click **Square**.

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

3 In the **Side length** text field, type `0.05`.

4 Locate the **Position** section. From the **Base** list, choose **Center**.

5 In the **xw** text field, type `-0.4`.

6 In the **yw** text field, type `0.2`.

Work Plane 1 (wp1)>Square 2 (sq2)

1 In the **Work Plane** toolbar, click **Square**.

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

3 In the **Side length** text field, type `0.05`.

4 Locate the **Position** section. From the **Base** list, choose **Center**.

5 In the **xw** text field, type `-0.4`.

6 In the **yw** text field, type `-0.2`.

Revolve 1 (rev1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Revolve**.

Sphere 1 (sph1)

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 `1.3`.

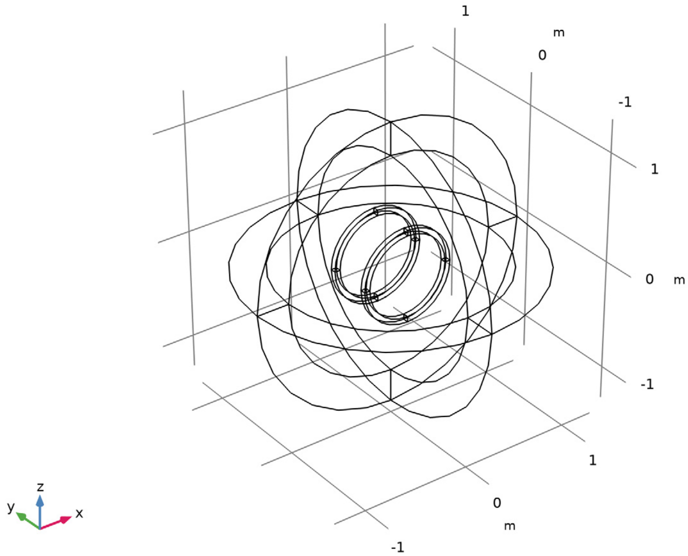
4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`0.3`

5 Click **Build All Objects**.

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

7 The geometry is now complete. To see its interior, click the **Wireframe Rendering** button in the **Graphics** toolbar.



Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **sph1**, select Point 4 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Find the **End vertex** subsection. Select the **Activate Selection** toggle button.

5 On the object **sph1**, select Point 9 only.

DEFINITIONS

Next, define the 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 **Geometry** section.

3 From the **Type** list, choose **Spherical**.

4 Select Domains 1–4 and 8–11 only.

Hide for Physics 1

1 In the **Model Builder** window, right-click **View 1** and choose **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 6 and 10 only.

MATERIALS

Define the materials for the model.

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.

By default, the first material added is applied on all domains.

Add another material for the coil domains.

MATERIALS

Coil Insulator

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

2 In the **Settings** window for **Material**, type `Coil Insulator` in the **Label** text field.

3 Select Domains 6 and 7 only.

4 Locate the **Material Contents** section. 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	6e7[S/m]	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

MAGNETIC FIELDS (MF)

Coil 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Magnetic Fields (mf)** and choose the domain setting **Coil**.

2 Select Domain 6 only.

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

4 From the **Conductor model** list, choose **Homogenized multiturn**.

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

6 In the I_{coil} text field, type `I0`.

In order to specify the direction of the wires in the circular coil, use the **Coil Geometry** subfeature to select a group of edges forming a circle. The path of the wires will be automatically computed from the geometry of the selected edges. For the best results, the radius of the circular edges selected should be close to the average radius of the coil.

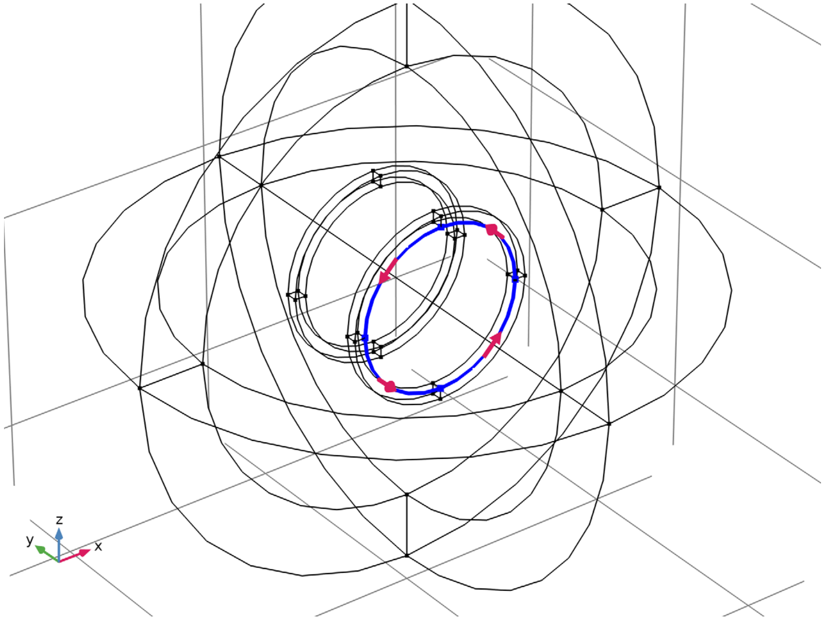
Coil Geometry 1

1 In the **Model Builder** window, click **Coil Geometry 1**.

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

3 Click **Clear Selection**.

4 Select Edges 25, 26, 46, and 49 only.



Now set up the second coil in the same way.

Coil 2

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

2 Select Domain 7 only.

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

4 From the **Conductor model** list, choose **Homogenized multiturn**.

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

6 In the I_{coil} text field, type `I0`.

Coil Geometry 1

1 In the **Model Builder** window, click **Coil Geometry 1**.

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

3 Click **Clear Selection**.

4 Select Edges 30, 31, 72, and 75 only.

MAGNETIC FIELDS, CURRENTS ONLY (MFCO)

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

Conductor 1

1. In the **Physics** toolbar, click **Domains** and choose **Conductor**.
2. Select Domain 6 only.

Terminal 1

1. In the **Model Builder** window, expand the **Conductor 1** node, then click **Terminal 1**.
2. Select Boundary 13 only.
3. In the **Settings** window for **Terminal**, locate the **Terminal** section.
4. In the I_0 text field, type `10*I0`.

Conductor 2

1. In the **Physics** toolbar, click **Domains** and choose **Conductor**.
2. Select Domain 7 only.

Terminal 1

1. In the **Model Builder** window, expand the **Conductor 2** node, then click **Terminal 1**.
2. Select Boundary 20 only.
3. In the **Settings** window for **Terminal**, locate the **Terminal** section.
4. In the I_0 text field, type `10*I0`.

MESH 1

Edge 1

1. In the **Mesh** toolbar, click **Boundary** and choose **Edge**.
2. Select Edge 40 only.

Distribution 1

1. Right-click **Edge 1** and choose **Distribution**.
2. In the **Settings** window for **Distribution**, locate the **Distribution** section.
3. In the **Number of elements** text field, type `50`.

Free Tetrahedral 1

1. In the **Mesh** toolbar, click **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 Select Domains 5–7 only.

Size 1

1 Right-click **Free Tetrahedral 1** and choose **Size**.

2 Select Domains 6 and 7 only.

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

4 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 `0.05`.

Swept 1

In the **Mesh** toolbar, click **Swept**.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

2 Right-click **Distribution 1** and choose **Build All**.

STUDY 1

Stationary 2

In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

Step 1: Stationary

1 In the **Model Builder** window, 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 **Magnetic Fields, Currents Only (mfco)**.

Step 2: Stationary 2

1 In the **Model Builder** window, click **Step 2: Stationary 2**.

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 **Magnetic Fields (mf)**.

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

Add a selection to the computed dataset to exclude the outer boundaries.

DEFINITIONS

Coils

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

2 Select Domains 6 and 7 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 1** and choose **Rename**.

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

7 Click **OK**.

Now add the plots.

RESULTS

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

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 **Boundary**.

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

Magnetic Flux Density, MF

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Magnetic Flux Density, MF` in the **Label** text field.

Slice 1

1 Right-click **Magnetic Flux Density, MF** 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 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Magnetic Fields>Magnetic>mf.normB - Magnetic flux density norm - T**.

6 In the **Magnetic Flux Density, MF** toolbar, click **Plot**.

Arrow Volume 1

1 In the **Model Builder** window, right-click **Magnetic Flux Density, 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.Hx,mf.Hy,mf.Hz - Magnetic field**.

3 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type 24.

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

5 Find the **z grid points** subsection. In the **Points** text field, type 1.

6 Locate the **Coloring and Style** section. Select the **Scale factor** check box.

7 In the associated text field, type 25.

8 In the **Magnetic Flux Density, MF** toolbar, click **Plot**.

To make the coil look like a solid object, can add a surface plot on its boundaries.

Surface 1

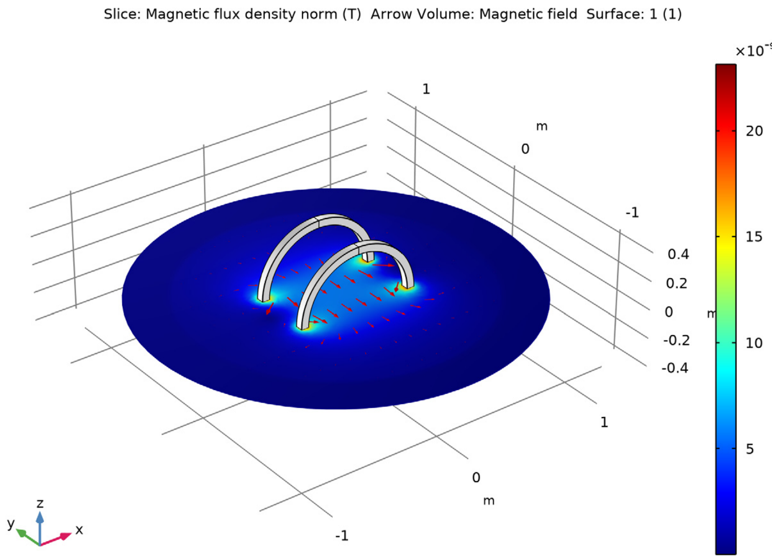
1 Right-click **Magnetic Flux Density, MF** and choose **Surface**.

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

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

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

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



Next, compare the results of By and Byy calculated from the two interfaces.

Comparison of By

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

2 In the **Settings** window for **1D Plot Group**, type `Comparison of By` in the **Label** text field.

Line Graph 1

1 Right-click **Comparison of By** and choose **Line Graph**.

2 Select Edge 40 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `mf.By`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `y`.

7 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

8 Click to expand the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`Magnetic Fields interface`

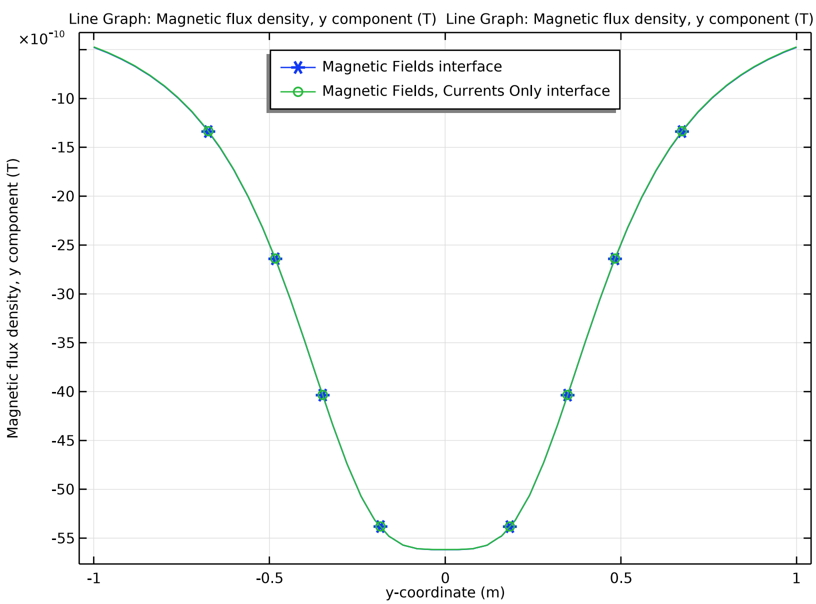
Line Graph 2

1. Right-click **Line Graph 1** and choose **Duplicate**.
2. In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `mfco.By`.
4. Locate the **Legends** section. In the table, enter the following settings:

Legends
`Magnetic Fields, Currents Only interface`

Comparison of By

1. In the **Model Builder** window, click **Comparison of By**.
2. In the **Settings** window for **1D Plot Group**, locate the **Legend** section.
3. From the **Position** list, choose **Upper middle**.
4. In the **Comparison of By** toolbar, click **Plot**.



Comparison of Byy

1 Right-click **Comparison of By** and choose **Duplicate**.

2 In the **Model Builder** window, click **Comparison of By 1**.

3 In the **Settings** window for **1D Plot Group**, type `Comparison of Byy` in the **Label** text field.

4 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Line Graph 1

1 In the **Model Builder** window, click **Line Graph 1**.

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

3 In the **Expression** text field, type `d(laginterp(2,mf.By),y)`.

The `mf` interface is using the Curl shape functions and the higher order spatial derivative is not available in postprocessing. In this case, use the `laginterp` operator.

Line Graph 2

1 In the **Model Builder** window, click **Line Graph 2**.

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

3 In the **Expression** text field, type `d(mfco.By,y)`.

The `mfco` interface is using secondary order Lagrange shape functions and second derivative is available. The curves of `Byy` can be improved by using cubic elements.

4 In the **Comparison of Byy** toolbar, click **Plot**.