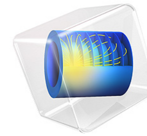
Created in COMSOL Multiphysics 5.6



Permanent Magnet

Introduction

This example shows how to model the magnetic field surrounding a permanent magnet. It also computes the force with which it acts on a nearby iron rod. Thanks to the symmetry of the geometry and the antisymmetry of the magnetic field, only one fourth of the geometry needs to be modeled.

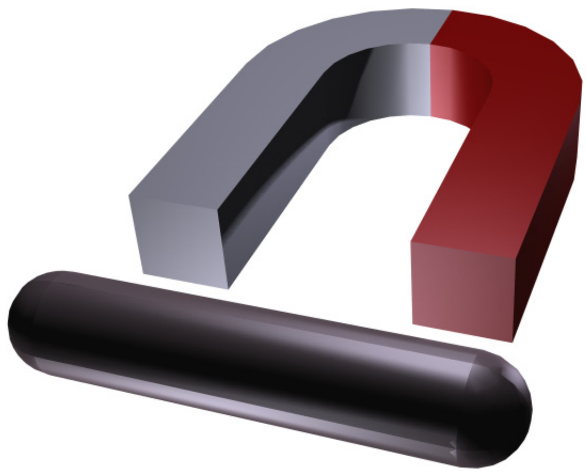


Figure 1: A full 3D view of the geometry. Left-right and top-down symmetry is used to minimize the problem size.

Model Definition

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

$$\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M})$$

together with the equation

$$\nabla \cdot \mathbf{B} = 0$$

you can derive an equation for V_m,

$$-\nabla\cdot(\mu_0\nabla V_m-\mu_0\mathbf{M}_0)=0$$

The model uses this equation by selecting the Magnetic Fields, No Currents interface from the AC/DC Module.

Boundary Conditions

The magnetic field is symmetric with respect to the *xy*-plane and antisymmetric with respect to the *xz*-plane. These planes therefore serve as exterior boundaries to the geometry.

On the symmetry plane, the magnetic field is tangential to the boundary. This is described by the Magnetic Insulation condition:

$$\mathbf{n}\cdot(\mu_0\nabla V_m-\mu_0\mathbf{M}_0)=\mathbf{n}\cdot\mathbf{B}=0$$

On the antisymmetry plane, the magnetic field is perpendicular to the boundary. This condition is represented by a constant magnetic scalar potential. The model uses the Zero Magnetic Scalar Potential condition.

If the air box is sufficiently large, the boundary condition used on its remaining exterior boundaries has little influence on the field in the vicinity of the magnet. Although an infinite element domain would give the very best results, this model uses the magnetic insulation condition for convenience.

Results and Discussion

The force on the rod is calculated internally as an integral of the surface stress tensor over all boundaries of the rod. The expression for the stress tensor reads

$$\mathbf{n}_1T_2=-\frac{1}{2}(\mathbf{H}\cdot\mathbf{B})\mathbf{n}_1+(\mathbf{n}_1\cdot\mathbf{H})\mathbf{B}^T$$

where $\mathbf{n}_1$ is the boundary normal pointing out from the rod and T_2 the stress tensor of air. The integration gives 1.52 N, which corresponds to one quarter of the rod. The actual force on the rod is therefore four times this value, or 6.1 N.

Application Library path: `ACDC_Module/Magnetostatics/permanent_magnet`

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

Import 1 (imp1)

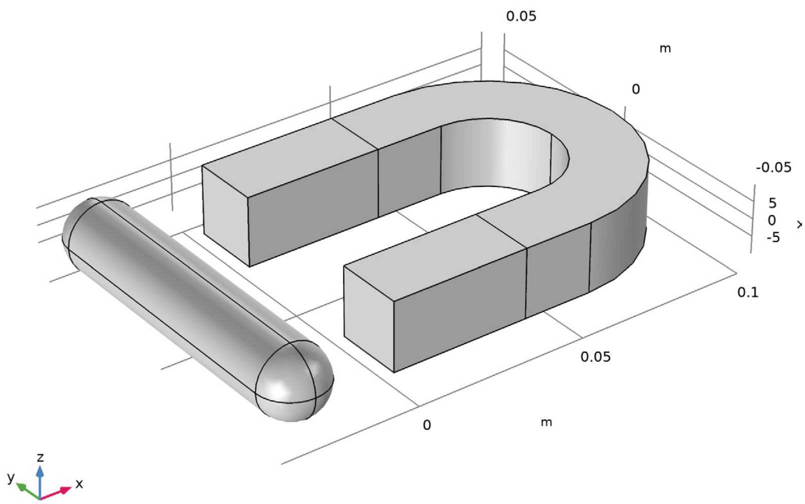
1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `permanent_magnet.mphbin`.

5 Click **Import**.



The imported geometry contains the permanent magnet and the rod that it is acting on. The following instructions show you how to create the air box and delete the part of the geometry that you do not want to include in the model.

Block 1 (blk1)

1 In 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 `0.25`.

4 In the **Depth** text field, type `0.1`.

5 In the **Height** text field, type `0.1`.

6 Locate the **Position** section. In the **x** text field, type `-0.1`.

7 Right-click **Block 1 (blk1)** and choose **Build Selected**.

The air box now covers only the parts of the magnet and the rod that you want to include in the model. Perform a Boolean geometry operation to get rid of the superfluous parts.

Compose 1 (co1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Compose**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

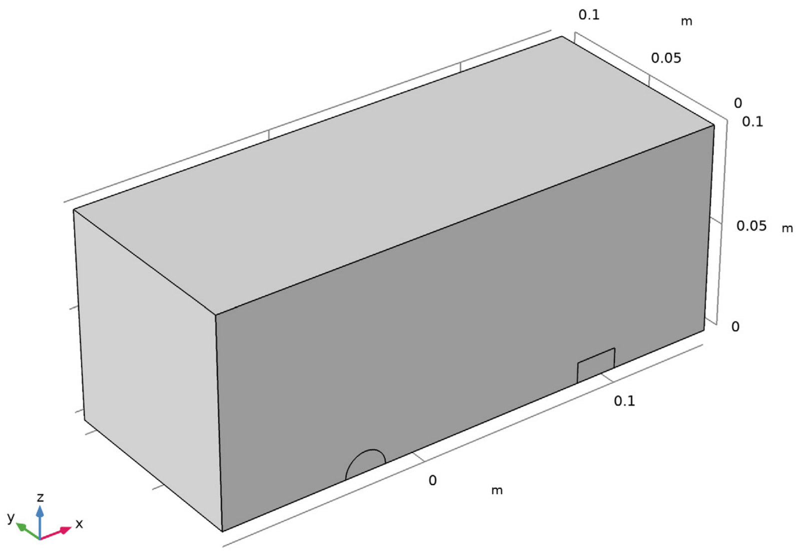
3 In the **Settings** window for **Compose**, locate the **Compose** section.

4 In the **Set formula** text field, type `blk1+imp1*blk1`.

5 Click **Build All Objects**.

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

The geometry now contains the air volume and one fourth of the imported objects.



MATERIALS

Iron

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 `Iron` in the **New label** text field.

4 Click **OK**.

5 Select Domains 2 and 4 only.

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

7 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	4000	1	Basic

Air

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 Right-click **Material 2 (mat2)** and choose **Rename**.

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

4 Click **OK**.

5 Select Domain 1 only.

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

7 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

MAGNETIC FIELDS, NO CURRENTS (MFNC)

Magnetic Flux Conservation 2

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Magnetic Fields, No Currents (mfnc)** and choose **Magnetic Flux Conservation**.

2 Select Domain 3 only.

3 In the **Settings** window for **Magnetic Flux Conservation**, locate the **Constitutive Relation B-H** section.

4 From the **Magnetization model** list, choose **Magnetization**.

5 Specify the **M** vector as

`750[kA/m]`	x

0	y
0	z

All exterior boundaries are magnetically insulated by default. Use the **Zero Magnetic Scalar Potential** condition on those boundaries where antisymmetry holds.

Zero Magnetic Scalar Potential 1

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

2 Select Boundaries 2, 8, and 24 only.

Next, add a force computation on the rod.

Force Calculation 1

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

2 Select Domain 2 only.

3 In the **Settings** window for **Force Calculation**, locate the **Force Calculation** section.

4 In the **Force name** text field, type `rod`.

MESH 1

Free Tetrahedral 1

To get an accurate force computation, you need a particularly fine mesh on the rod. It also makes sense to use a fine mesh in the magnet and its iron core, as this is where the magnetic field will be the strongest.

In the **Mesh** toolbar, click **Free Tetrahedral**.

Size

1 In the **Model Builder** window, click **Size**.

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

3 From the **Predefined** list, choose **Fine**.

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 **Geometric entity level** list, choose **Domain**.

4 Select Domains 2–4 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 `0.0025`.

8 Click **Build All**.

STUDY 1

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

RESULTS

3D Plot Group 1

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

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 **Component 1 (comp1)>Magnetic Fields, No Currents>Magnetic>mfnc.normB - Magnetic flux density norm - T**.

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 `0.005`.

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

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

The plot shows the magnitude of the flux density just above the symmetry plane. Add an arrow plot to see its direction.

Arrow Volume 1

1 In the **Model Builder** window, right-click **3D Plot Group 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, No Currents>Magnetic>mfnc.Bx,...,mfnc.Bz - Magnetic flux density**.

3 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `100`.

4 Find the **y grid points** subsection. In the **Points** text field, type `50`.

5 Find the **z grid points** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type `0.0051`.

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

The plot shows only the quarter of the geometry used for the computation.

Introduce additional mirror datasets to plot the solution in the complete geometry.

Symmetry Condition

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, type `Symmetry Condition` in the **Label** text field.

3 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

Antisymmetry Condition

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, type `Antisymmetry Condition` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Symmetry Condition**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

5 Click to expand the **Advanced** section. From the **Vector transformation** list, choose **Antisymmetric**.

3D Plot Group 1

1 In the **Model Builder** window, click **3D Plot Group 1**.

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

3 From the **Dataset** list, choose **Antisymmetry Condition**.

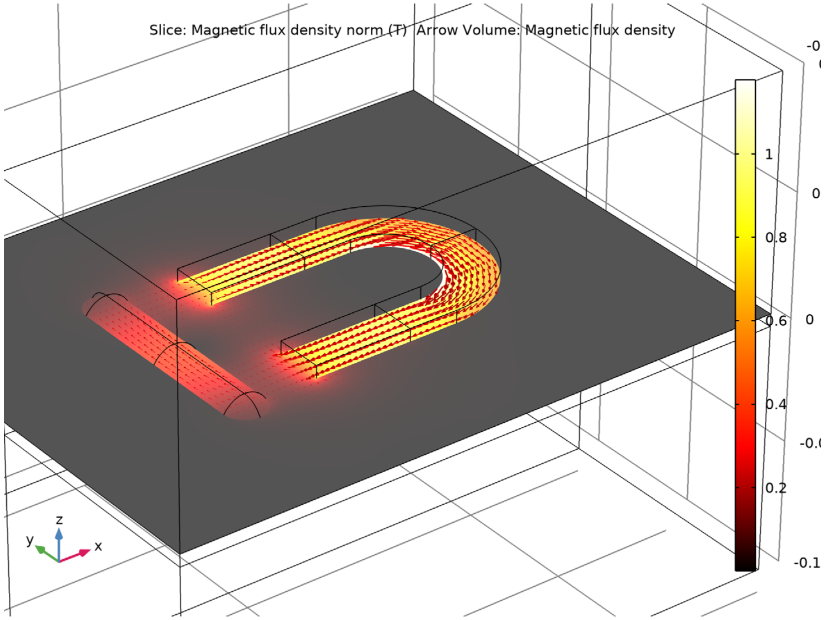
4 In the **3D Plot Group 1** toolbar, click **Plot**.

Arrow Volume 1

1 In the **Model Builder** window, click **Arrow Volume 1**.

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, No Currents>Magnetic>mfnc.Bx,...,mfnc.Bz - Magnetic flux density**.

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



3D Plot Group 1

Finally, use Global Evaluation to evaluate the force on the rod.

Global Evaluation 1

1 In 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 **Component 1 (comp1)>Magnetic Fields, No Currents>Mechanical>Electromagnetic force - N>mfnc.Forcex_rod - Electromagnetic force, x component**.

The variable containing the force on a quarter of the rod will be added to the **Expressions** table. Multiply this value by 4 to compute the total force on the rod.

3 In the **Expression** column, change the expression to `mfnc.Forcex_rod*4`.

4 In the **Description** column, write `Total force on the rod`.

5 Click **Evaluate**.

The total force on the rod evaluates to 6.1 N.