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.

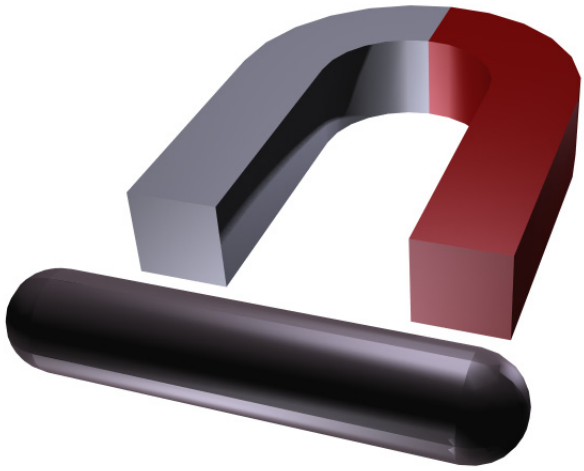


Figure 1: A full 3D view of the geometry. Left-right and top-down symmetry is used to minimize the problem size.

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 (see Infinite Elements in the *AC/DC Module User's Guide*) 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

$$\mathbf{n}_1 T_2 = -\frac{1}{2}(\mathbf{H} \cdot \mathbf{B})\mathbf{n}_1 + (\mathbf{n}_1 \cdot \mathbf{H})\mathbf{B}^T$$

where $\mathbf{n}_1$ is the boundary normal pointing out from the rod and T_2 the stress tensor of air. The integration gives 1.53 N, which corresponds to one quarter of the rod. The actual force on the rod is therefore four times this value, or 6.11 N.

 ©2012 COMSOL

Model Library path: `ACDC_Module/Magnetostatics/permanent_magnet`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **AC/DC>Magnetic Fields, No Currents (mfnc)**.

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

6 Click **Finish**.

GEOMETRY 1

Import 1

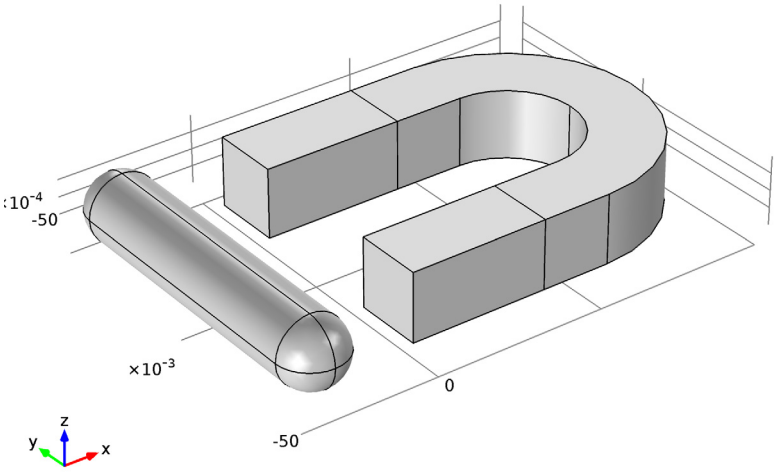
1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Import**.

2 In the **Import** settings window, locate the **Import** section.

3 Click the **Browse** button.

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

5 Click the **Import** button.



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

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Block**.

2 In the **Block** settings window, locate the **Size and Shape** section.

3 In the **Width** edit field, type `0.25`.

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

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

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

7 Click the **Build All** button.

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

1 Right-click **Geometry 1** and choose **Boolean Operations>Compose**.

 ©2012 COMSOL

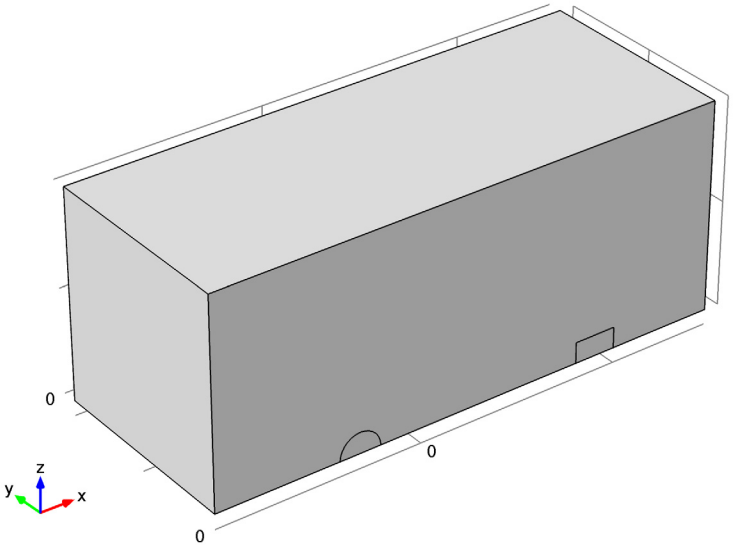
2 Select the objects **imp1** and **blk1** only.

3 In the **Compose** settings window, locate the **Compose** section.

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

5 Click the **Build All** button.

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



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

MATERIALS

Material 1

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

2 Right-click **Material 1** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Iron` in the **New name** edit field.

4 Click **OK**.

5 Select Domains 2 and 4 only.

6 In the **Material** settings window, locate the **Material Contents** section.

7 In the table, enter the following settings:

Property	Name	Value
Relative permeability	mur	4000

Material 2

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

2 Right-click **Material 2** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Air` in the **New name** edit field.

4 Click **OK**.

5 Select Domain 1 only.

6 In the **Material** settings window, locate the **Material Contents** section.

7 In the table, enter the following settings:

Property	Name	Value
Relative permeability	mur	1

MAGNETIC FIELDS, NO CURRENTS

Magnetic Flux Conservation 2

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

2 Select Domain 3 only.

3 In the **Magnetic Flux Conservation** settings window, locate the **Magnetic Field** section.

4 From the **Constitutive relation** list, choose **Magnetization**.

5 In the $\mathbf{M}$ table, enter the following settings:

750[kA/m]	x
0	y
0	z

Zero Magnetic Scalar Potential 1

All exterior boundaries are magnetically insulated by default. Use the Zero potential condition on those boundaries where antisymmetry holds.

1 In the **Model Builder** window, right-click **Magnetic Fields, No Currents** and choose **Zero Magnetic Scalar Potential**.

2 Select Boundaries 2, 8, and 24 only.

 ©2012 COMSOL

Force Calculation 1

Next, add a force computation on the rod.

1 Right-click **Magnetic Fields, No Currents** and choose **Force Calculation**.

2 Select Domain 2 only.

3 In the **Force Calculation** settings window, locate the **Force Calculation** section.

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

MESH 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 highest.

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

Size

1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.

2 In the **Size** settings window, locate the **Element Size** section.

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

Size 1

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

2 In the **Size** settings window, 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 edit field, type `0.0025`.

8 Click the **Build All** button.

STUDY 1

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

2 In the **Study** settings window, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

4 Click the **Compute** button.

RESULTS

3D Plot Group 1

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

2 Right-click **3D Plot Group 1** and choose **Slice**.

3 In the **Slice** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Magnetic Fields, No Currents>Magnetic>Magnetic flux density norm (mfnc.normB)**.

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

5 From the **Entry method** list, choose **Coordinates**.

6 In the **z-coordinates** edit field, type `0.005`.

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

8 Click the **Plot** button.

You are now looking at the magnitude of the flux density just above the symmetry plane. With an arrow plot, you can see the direction too.

9 In the **Model Builder** window, right-click **3D Plot Group 1** and choose **Arrow Volume**.

10 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Magnetic Fields, No Currents>Magnetic>Magnetic flux density (mfnc.Bx,...,mfnc.Bz)**.

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

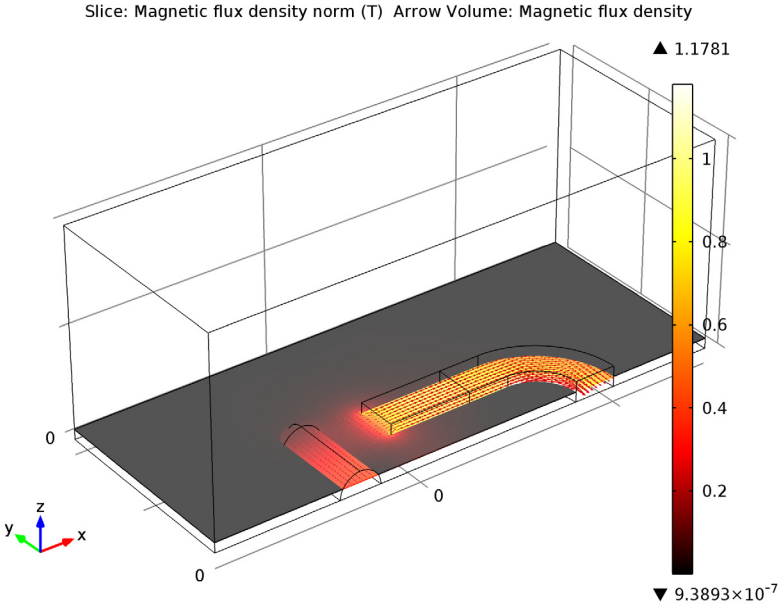
12 Find the **y grid points** subsection. In the **Points** edit field, type `50`.

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

14 In the **Coordinates** edit field, type `0.0051`.

 ©2012 COMSOL

15 Click the **Plot** button.



Derived Values

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

1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Global Evaluation**.

2 In the **Global Evaluation** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Magnetic Fields, No Currents>Mechanical>Electromagnetic force>Electromagnetic force, x component (mfnc.Forcex_rod)**.

3 Click the **Evaluate** button.

The force on a quarter of the rod evaluates to 1.53 N.

 ©2012 COMSOL