Model created in COMSOL Multiphysics 6.4

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.

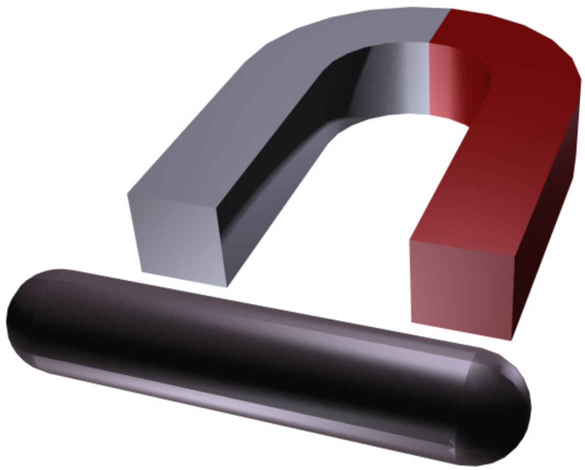


Figure 1: A full 3D view of the geometry. Left-right and top-down symmetry is used to minimize the problem size.

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

$$\mathbf{B} = \mu_0 \mu_\mathrm{rec} \mathbf{H} + \mathbf{B}_\mathrm{r}$$

where $\mathbf{B}_\mathrm{r}$ is the remanent flux density, and together with the equation

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

you can derive an equation for V_m,

$$-\nabla \cdot (\mu_0 \mu_{rec} \nabla V_m - \mathbf{B}_r) = 0$$

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:

$$\mathbf{n} \cdot (\mu_0 \mu_{rec} \nabla V_m - \mathbf{B}_r) = \mathbf{n} \cdot \mathbf{B} = 0$$

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

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

where $\mathbf{n}_1$ is the boundary normal pointing out from the rod and T_2 the stress tensor of air. The integration gives 3.64 N, which corresponds to one quarter of the rod. The actual force on the rod is therefore four times this value, or 14.6 N.

Application Library path: `ACDC_Module/Introductory_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)

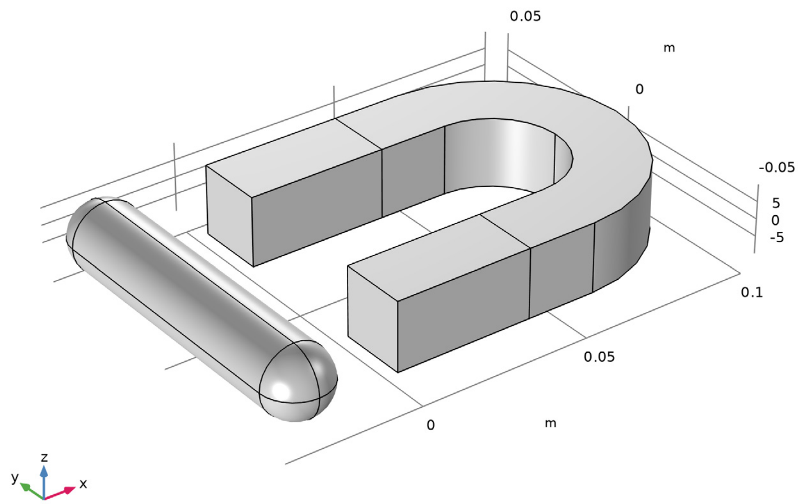
1 In the **Geometry** toolbar, click **Import**.

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

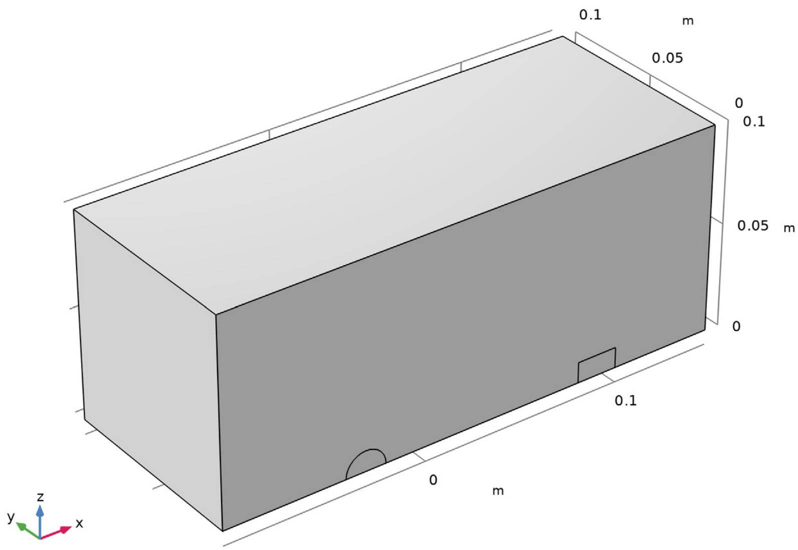
3 In the **Settings** window for **Compose**, locate the **Compose** section.

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.



MAGNETIC FIELDS, NO CURRENTS (MFNC)

Magnetic Flux Conservation in Solids 1

1 In the **Physics** toolbar, click **Domains** and choose **Magnetic Flux Conservation in Solids**.

2 Select Domains 2–4 only.

MATERIALS

Iron

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 `Iron` in the **Label** text field.

3 Select Domains 2 and 4 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	4000	1	Basic

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **AC/DC** > **Hard Magnetic Materials** > **Sintered NdFeB Grades (Chinese Standard)** > **N54 (Sintered NdFeB)**.

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

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

MATERIALS

N54 (Sintered NdFeB) (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 Click **Paste Selection**.

3 In the **Paste Selection** dialog, type 3 in the **Selection** text field.

4 Click **OK**.

MAGNETIC FIELDS, NO CURRENTS (MFNC)

Magnet 1

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

2 Select Domain 3 only.

North 1

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

2 Select Boundary 17 only.

South 1

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

2 Select Boundary 12 only.

All exterior boundaries are magnetically insulated by default. Use the **Symmetry Plane** condition on those boundaries where antisymmetry holds.

Symmetry Plane 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Symmetry Plane**.
2. In the **Model Builder** window, click **Symmetry Plane 1**.
3. Select Boundaries 2, 8, and 24 only.
4. In the **Settings** window for **Symmetry Plane**, locate the **Symmetry Plane** section.
5. From the **Symmetry type for the magnetic field** list, choose **Antisymmetry**.

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`.

STUDY 1

Default mesh is sufficient in this case where the force is to be computed only on the rod, and we can directly go to the solution step.

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

RESULTS

3D Plot Group 1

In the **Results** toolbar, click **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 **GrayBody**.
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.

Magnetic Flux Density

1 In the **Model Builder** window, under **Results** click **3D Plot Group 1**.
2 In the **Settings** window for **3D Plot Group**, type `Magnetic Flux Density` in the **Label** text field.

Arrow Volume 1

1 Right-click **Magnetic Flux Density** 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 **Magnetic Flux Density** 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. Find the **Space variables** subsection. From the **Vector transformation** list, choose **Antisymmetric**.

Magnetic Flux Density

1 In the **Model Builder** window, under **Results** click **Magnetic Flux Density**.

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

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

4 In the **Magnetic Flux Density** 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 **Magnetic Flux Density** toolbar, click **Plot**.

Add Global Evaluation under Derived Values to statically store the force on the rod for the default mesh.

Force on Rod (Default Mesh)

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** > **Total electromagnetic force, object rod - N** > **mfnc.Forcex_rod - Total electromagnetic force, object rod, component x**.

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, type `Total force on the rod`.

5 In the **Label** text field, type `Force on Rod (Default Mesh)`.

6 Click **Evaluate**.

The total force on the rod evaluates to 15 N.

Now compute the force also on the horse-shoe magnet.

MAGNETIC FIELDS, NO CURRENTS (MFNC)

Force Calculation 2

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

2 Select Domains 3 and 4 only.

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

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

Boundary Force Calculation 1

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

2 Select Boundaries 1, 4, 5, and 25 only.

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

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

MESH 1

Build the default mesh.

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

For the magnet we need a finer mesh near to its sharp corners. Such a mesh can be specified by editing the default mesh.

2 Right-click **Component 1 (comp1)** > **Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size 2

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

2 In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** click **Size 2**.

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

4 Click **Clear Selection**.

5 Select Edges 21, 24–26, and 28 only.

6 Click to expand the **Element Size Parameters** section. Locate the **Element Size** section. Click the **Custom** button.

7 Locate the **Element Size Parameters** section.

8 Select the **Maximum element size** checkbox. In the associated text field, type `0.3[mm]`.

Size 1

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

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

3 Select the **Maximum element growth rate** checkbox. In the associated text field, type `1.1`.

Boundary Layer Properties 2

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Mesh 1** > **Boundary Layers 1** node, then click **Boundary Layer Properties 2**.

2 Select Boundaries 6, 9–13, 15, 16, 18, and 20–23 only.

3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

4 In the **Thickness adjustment factor** text field, type `1`.

STUDY 1

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

RESULTS

Force Calculation Results

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Force Calculation Results` in the **Label** text field.

Global Evaluation 1

1 Right-click **Results** and choose **Global Evaluation**.

Add the forces computed from the Force Calculation features, multiply the relevant ones by 4 to account for symmetries, and compute the forces on the rod and the horseshoe magnet.

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** > **Total electromagnetic force, object rod - N** > **mfnc.Forcex_rod - Total electromagnetic force, object rod, component x**.

3 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Magnetic Fields, No Currents** > **Mechanical** > **Total electromagnetic force, object horseShoeMagnet - N** > **mfnc.Forcex_horseShoeMagnet - Total electromagnetic force, object horseShoeMagnet, component x**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`4*mfnc.Forcex_rod`	N	Total electromagnetic force, rod
`4* mfnc.Forcex_horseShoeMagnet`	N	Total electromagnetic force, horse shoe magnet

5 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Magnetic Fields, No Currents** > **Mechanical** > **Total electromagnetic force, object exteriorBoundaries - N** > **mfnc.Forcex_exteriorBoundaries - Total electromagnetic force, object exteriorBoundaries, component x**.

6 In the **Force Calculation Results** toolbar, click **Evaluate**.

Appreciate that the force on the rod is changing just slightly, that the force on the horse-shoe magnet is approximately minus the force on the rod, and that the effect of exterior boundaries is negligible.

FORCE CALCULATION RESULTS

Go to the **Force Calculation Results** window.