COMSOL MULTIPHYSICS®

Permanent Magnet

SOLVED WITH COMSOL MULTIPHYSICS 3.5a

© COPYRIGHT 2008. All right reserved. No part of this documentation may be photocopied or reproduced in any form without prior written consent from COMSOL AB. COMSOL, COMSOL Multiphysics, COMSOL Reaction Engineering Lab, and FEMLAB are registered trademarks of COMSOL AB. Other product or brand names are trademarks or registered trademarks of their respective holders.

COMSOL

Permanent Magnet

Introduction

In magnetostatic problems, where no currents are present, the problem can be solved using a scalar magnetic potential. This model illustrates this technique for a horse shoe-shaped permanent magnet placed near an iron rod.

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,

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

Results and Discussion

The force on the rod is calculated by integrating 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 148 N, which corresponds to one quarter of the rod. The actual force on the rod is therefore four times this value, 592 N.

Model Library path: `ACDC_Module/Tutorial_Models/rod_and_permanent_magnet`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. Open the Model Navigator.
2. On the **New** page, select **3D** from the **Space dimension** list.
3. From the **Application Modes** list, select **AC/DC Module>Statics, Magnetic>Magnetostatics, No Currents**. Click **OK**.

OPTIONS AND SETTINGS

1. From the **Options** menu, select **Constants**.

2 In the **Constants** dialog box enter the following names, expressions, and (optionally) descriptions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
murFe	5000	Relative permeability of iron
Mpre	750000	Magnetization of magnet (A/m)

GEOMETRY MODELING

You can use symmetries to reduce the size of the model. Only one arm of the magnet needs to be included in the model, and a horizontal cut can also be made. Thus, only one quarter of the magnet and rod will be modeled.

1 Create a work plane by opening the **Work-Plane Settings** dialog box and clicking **OK** to obtain the default work plane in the *xy*-plane.

2 In the **Axes/Grid Settings** dialog box, clear the **Auto** check box on the **Grid** tab and set both the grid **x spacing** and **y spacing** to 0.2.

3 Draw a rectangle R1 with corners at (0, 0.2) and (0.4, 0.4).

4 Draw another rectangle R2 with corners at (0.4, 0.2) and (0.6, 0.4).

5 Click the **Ellipse/Circle (Centered)** button on the Draw toolbar. Draw a circle C1 with center at (0.6, 0) and radius 0.2. Use the right mouse button to make sure a circle is obtained rather than an ellipse.

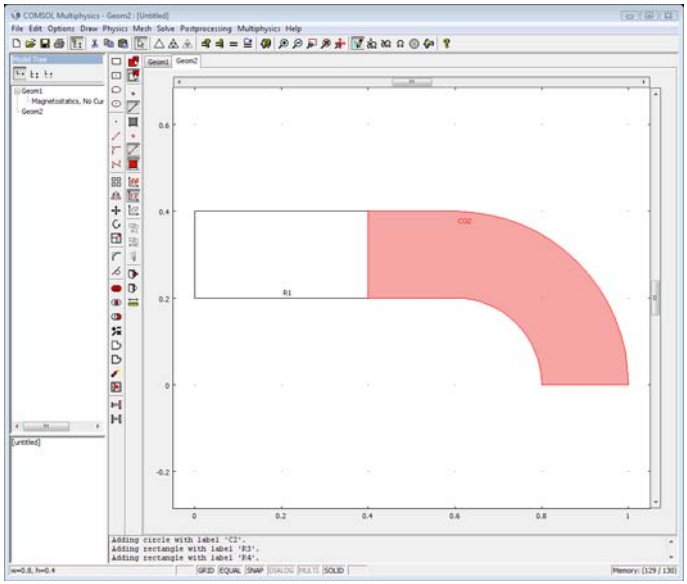
6 Draw another circle C2 also with center at (0.6, 0) but with radius 0.4.

7 Select both circles and click the **Difference** button to create the object CO1.

8 Draw a rectangle R3 with corners at (0.2, 0) and (0.6, 0.4), and a rectangle R4 with corners at (0.2, –0.4) and (1, 0).

9 Select CO1, R3, and R4 and click the **Difference** button to create the object CO2.

10 Select CO2 and R2 and click the **Union** button. Then click the **Delete Interior Boundaries** button.



11 Open the **Extrude** dialog box and select both R1 and CO2. Enter the **Distance** `0.1` and click **OK** to extrude the magnet.

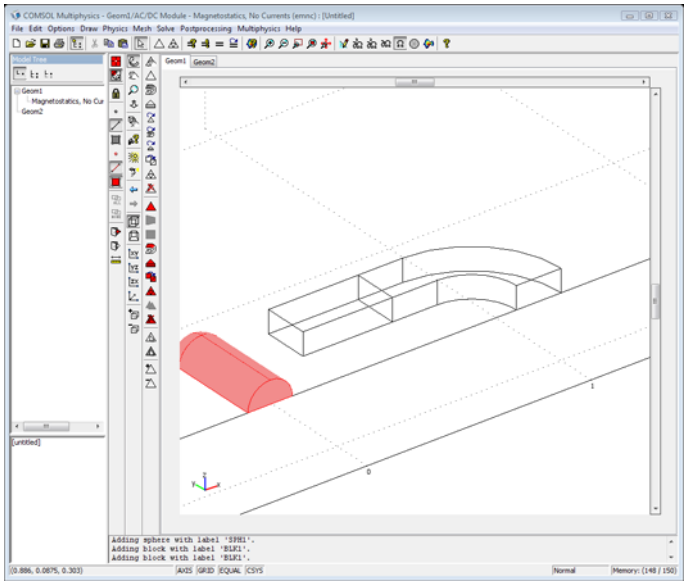
12 Create the rod by combining a cylinder and a sphere. Click the **Cylinder** button to open the **Cylinder** dialog box, and enter the **Radius** `0.1` and **Height** `0.4`. Set the **Axis base point** to (–0.3, 0, 0) and the **Axis direction vector** to (0, 1, 0). Click **OK** to create the cylinder.

13 Click the **Sphere** button and enter the **Radius** `0.1`. Set the **Axis base point** to (–0.3, 0.4, 0) then click **OK** to create the sphere.

14 Half of the rod should now be removed. This can be done by subtracting a block from the cylinder and sphere. Click the **Block** button and enter the **Length** coordinates (0.4, 0.6, 0.2). Set the **Axis base point** to (–0.5, –0.1, –0.2), then click **OK** to create a block.

15 Open the **Create Composite Object** dialog box, and type the formula `CYL1+SPH1-BLK1` in the **Set formula** edit field. Clear the **Keep interior boundaries** check box and then click **OK**.

16 Finally add a block surrounding the magnets and rod. In the **Block** dialog box use the **Length** coordinates (5, 2, 2) and the **Axis base point** (−2, 0, 0).



PHYSICS SETTINGS

Boundary Conditions

Along the boundaries far away from the magnet, the magnetic field should be tangential to the boundary as the flow lines should form closed loops around the magnet. The natural boundary condition is

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

Thus the magnetic field is made tangential to the boundary by a Neumann condition on the potential.

Along the symmetry boundary below the magnet, the magnetic field should also be tangential, and therefore you can apply the same Neumann condition there.

On the other hand, the magnetic field should be normal to the other symmetry boundary at the front in the figure above in order for the magnetic flow lines to form closed loops around the magnet. This means that the magnetic field is symmetric with respect to the boundary. This can be achieved by setting the potential to zero along

the boundary, and thus making the potential antisymmetric with respect to the boundary.

1 From the **Physics** menu, choose **Boundary Settings**.

2 Enter boundary conditions according to the following table; when done, click **OK**.

SETTINGS	BOUNDARIES 2, 8, 24	ALL OTHERS
Boundary condition	Zero potential	Magnetic insulation

Subdomain Settings

1 From the **Physics** menu, choose **Subdomain Settings**.

2 In the **Subdomain Settings** dialog box, enter coefficients according to the table below; when done, click **OK**.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2	SUBDOMAIN 3	SUBDOMAIN 4
Constitutive relation	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$	$\mathbf{B} = \mu_0\mathbf{H} + \mu_0\mathbf{M}$	$\mathbf{B} = \mu_0\mu_r\mathbf{H}$
μ_r	1	murFe		murFe
$\mathbf{M}$			Mpre 0 0	
Force variables		rod		

MESH GENERATION

1 Choose **Mesh>Free Mesh Parameters**.

2 From the **Predefined mesh sizes** list select **Fine**.

3 In order to make the mesh finer in the magnet and in the rod, click the **Subdomain** tab, then set the **Maximum element size** for Subdomains 2, 3, and 4 to 0.025.

4 Click **OK** to close the dialog box.

5 Click the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot shows a slice plot of the magnetic potential.

1 Click the **Plot Parameters** button on the Main toolbar.

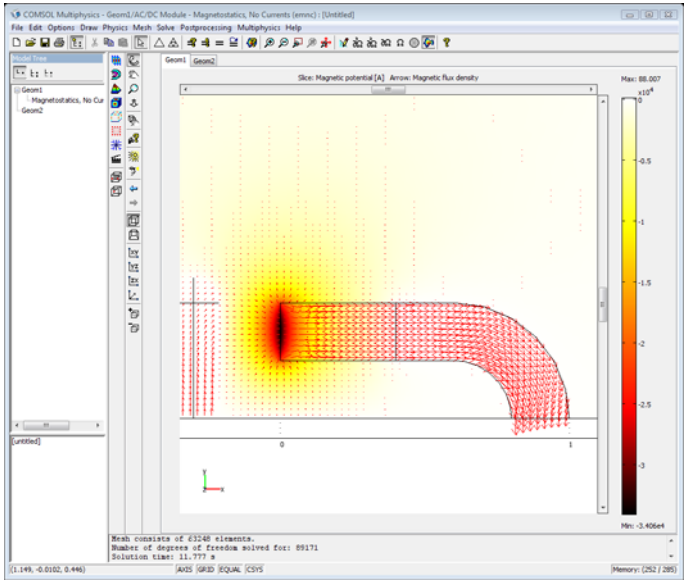
2 On the **General** page, select the **Slice** and **Arrow** check boxes in the **Plot type** area.

3 On the **Slice** page, set the **x levels** to 0 and the **y levels** to 0 in the **Slice positioning** area. For the **z levels** select **Vector with coordinates** and enter `0.05`. Select **Thermal** from the **Color table** list.

4 On the **Arrow** page, select the **Magnetic flux density** as **Arrow data**. Set the **x points** to 200 and the **y points** to 100 in the **Arrow positioning** area. For the **z points** select **Vector with coordinates** and enter `0.051`. Select **Arrow** from the **Arrow type** list.

5 Click **OK** to close the dialog and generate the plot.

6 Click the button **Goto XY View** and use the zoom tool to zoom in on the magnet and rod.



Force Calculation

To calculate the force on the rod, use the surface stress tensor, which is derived in "Electromagnetic Forces" on page 133 in the *AC/DC Module User's Guide*,

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

where $\mathbf{n}_1$ is the boundary normal pointing out from the rod and T_2 the stress tensor of air. The electromagnetic force variable `rod`, which was defined on the rod in the **Subdomain Settings** dialog box, generates the variables `rod_forcex_emnc`, `rod_forcey_emnc`, and `rod_forcez_emnc`, which are the forces on the rod in the x, y, and z directions, respectively.

To evaluate the force in the x direction, open the **Global Data Display** dialog box and enter the expression `rod_forcex_emnc`. The result is about 148 N. Because the model only includes one quarter of the rod, the actual force is four times this value, 592 N.