Axial Magnetic Bearing Using Permanent Magnets

Introduction

Permanent magnet bearings are used in turbo machinery, pumps, motors, generators, and flywheel energy storage systems, to mention a few application areas; contactless operation, low maintenance, and the ability to operate without lubrication are some key benefits compared to conventional mechanical bearings. This model illustrates how to calculate design parameters like magnetic forces and stiffness for an axial permanent magnet bearing.

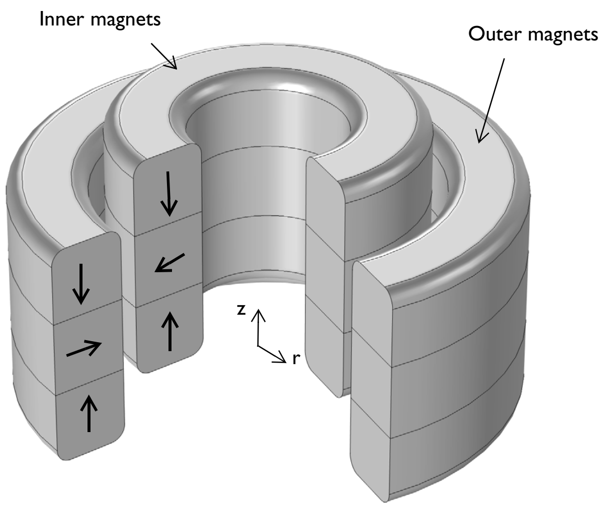


Figure 1: Model illustration of an axial magnetic bearing using permanent magnets. The black arrows show the magnetization direction of the permanent magnets.

Model Definition

Set up the problem in a 2D axisymmetric modeling space. Figure 1 shows a 3D view of the model with the magnetization directions of the magnets indicated. COMSOL Multiphysics calculates the total magnetic force on an object by integrating the vector expression

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

where $\mathbf{n}$ is the outward normal vector and T is the Maxwell stress tensor, over the object's outer boundaries.

The negative of the derivative of the total magnetic force with respect to the position is referred to as the magnetic stiffness. By this definition, the axial magnetic stiffness of the bearing is

$$k_z = -\frac{dF_z}{dz} \tag{1}$$

where F_z is the total axial magnetic force on the bearing. This model calculates the magnetic stiffness in the axial direction only; calculating the magnetic stiffness in the radial direction as well as the coupled stiffness coefficients requires a complete 3D model.

The model parameters are taken from Ref. 1.

Results

A steady-state study is performed to calculate the magnetic forces and the axial magnetic stiffness coefficient. Figure 2 shows the magnetic flux density norm and the magnetic vector potential for an axial displacement of the inner magnets of z = 40 mm. Figure 3 and Figure 4 illustrate the radial and axial component, respectively, of the magnetic force on the inner magnets as a function of axial displacement. Figure 5 displays the sensitivity of the axial magnetic force with respect to the axial displacement. The negative of this plot is the axial magnetic stiffness coefficient. Finally, Figure 6 shows the magnetic flux density norm in 3D at an axial displacement of 8 mm.

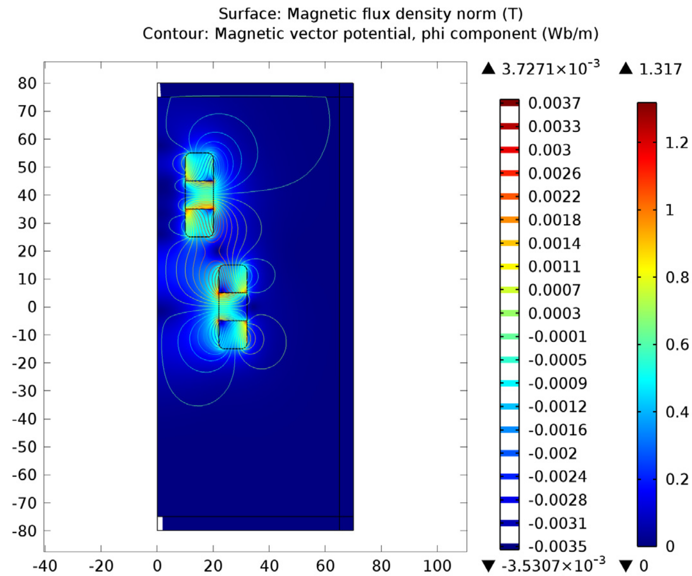


Figure 2: Magnetic flux density norm and magnetic vector potential for an axial displacement of the inner magnets of 40 mm.

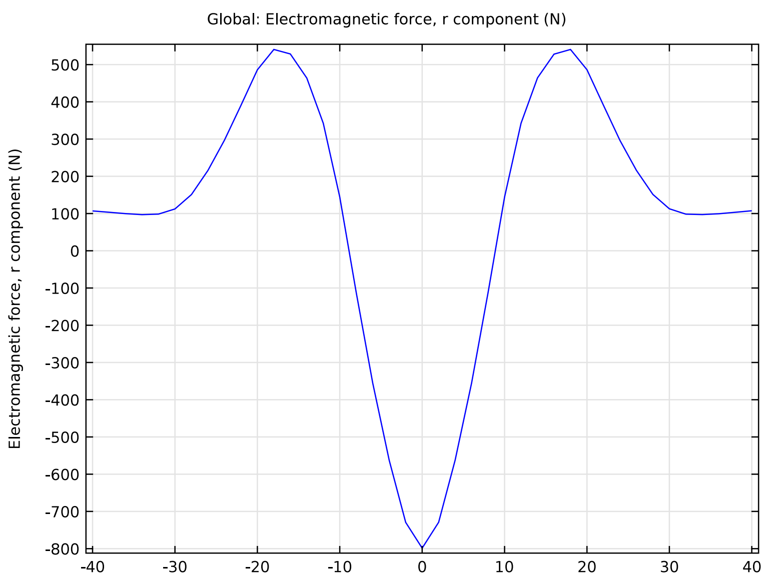


Figure 3: Radial component of the magnetic force versus axial displacement.

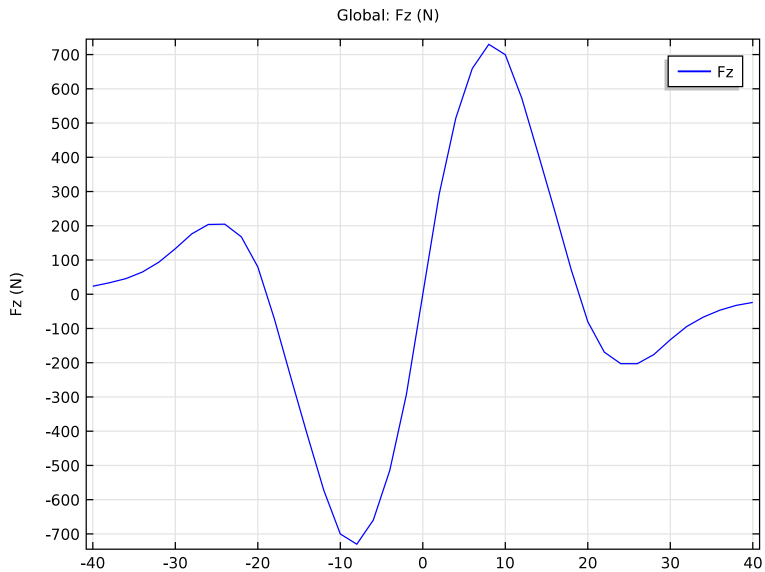


Figure 4: Axial component of the magnetic force versus axial displacement.

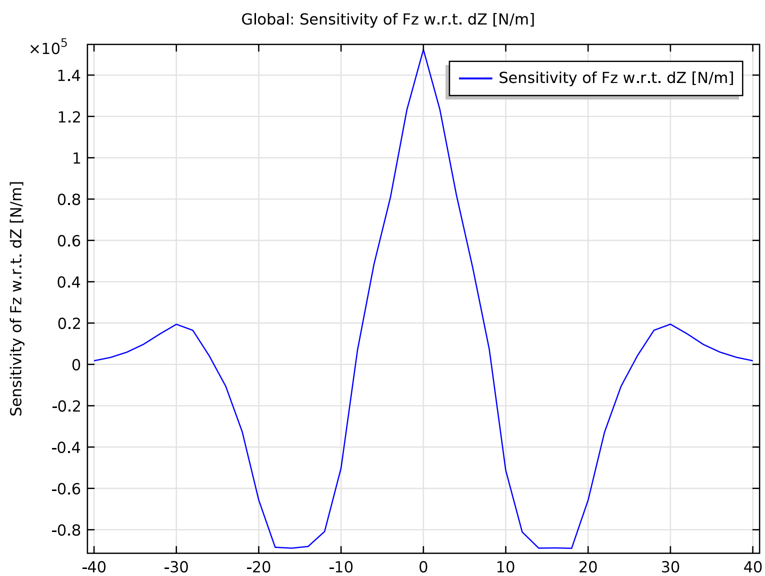


Figure 5: Sensitivity of the axial magnetic force with respect to axial displacement versus axial displacement. The negative of this quantity is the axial magnetic stiffness coefficient.

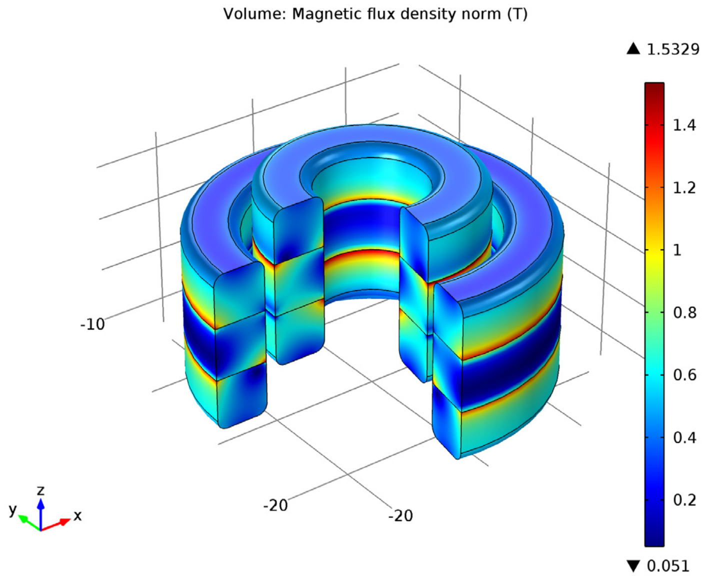


Figure 6: Magnetic flux density norm at an axial displacement of 8 mm.

Notes About the COMSOL Implementation

Use the Magnetic Fields physics to model the magnetic field. Add an Infinite Element Domain to model the open region of free space surrounding the magnets. Calculate the total magnetic force components with the Maxwell stress tensor method by adding a Force Calculation node in the inner magnet domains. Also, add Deformed Geometry and Sensitivity interfaces to calculate the axial magnetic stiffness coefficient as defined by Equation 1. With the Deformed Geometry interface you parameterize the axial displacement of the inner magnets. Then, you use the axial component of the magnetic force as a global objective and the axial displacement parameter as a global control variable for the Sensitivity interface to obtain the derivative dF_z/dz. Using a Parametric Sweep study node, you finally compute the axial magnetic stiffness as a function of the axial displacement.

Reference

1. R. Ravaud and G. Lemarquand, "Halbach Structures for Permanent Magnet Bearings," *Progress In Electromagnetics Research M*, vol. 14, pp. 236–277, 2010.

Model Library path: ACDC_Module/Motors_and_Actuators/axial_magnetic_bearing

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.
2 Click the **2D axisymmetric** button.
3 Click **Next**.
4 In the **Add physics** tree, select **AC/DC>Magnetic Fields (mf)**.
5 Click **Next**.
6 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.
7 Click **Finish**.

GLOBAL DEFINITIONS

Define all the necessary parameters here.

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
2 In the **Parameters** settings window, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Description
R1	10[mm]	Inner radius of inner magnet
R2	20[mm]	Outer radius of inner magnet
R3	22[mm]	Inner radius of outer magnet
R4	32[mm]	Outer radius of outer magnet
h0	10[mm]	Magnet height
Br	1[T]	Remanent flux density of magnet
dZ	0[mm]	Axial displacement

Later, you will use dZ as a global control variable for a Sensitivity physics and as the parameter of a Parametric Sweep node in order to compute the axial magnetic stiffness.

GEOMETRY 1

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

2 In the **Geometry** settings window, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Follow the instructions below to construct the model geometry.

Rectangle 1

1 Right-click **Model 1>Geometry 1** and choose **Rectangle**.

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

3 In the **Width** edit field, type `R2-R1`.

4 In the **Height** edit field, type `h0*3`.

5 Locate the **Position** section. In the **r** edit field, type `R1`.

6 In the **z** edit field, type `-h0/2-h0+dZ`.

7 Click the **Build Selected** button.

8 In the **Model Builder** window, click **Rectangle 1**.

9 In the **Rectangle** settings window, click to expand the **Layers** section.

10 In the table, enter the following settings:

Thickness (mm)
h0

11 Select the **Layers on top** check box.

12 Click the **Build Selected** button.

Rectangle 2

1 Right-click **Rectangle 1** and choose **Duplicate**.

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

3 In the **Width** edit field, type `R4-R3`.

4 Locate the **Position** section. In the **r** edit field, type `R3`.

5 In the **z** edit field, type `-h0/2-h0`.

6 Click the **Build Selected** button.

Rectangle 3

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

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

3 In the **Width** edit field, type 70.
4 In the **Height** edit field, type 160.
5 Locate the **Position** section. In the **z** edit field, type -80.
6 Locate the **Layers** section. In the table, enter the following settings:

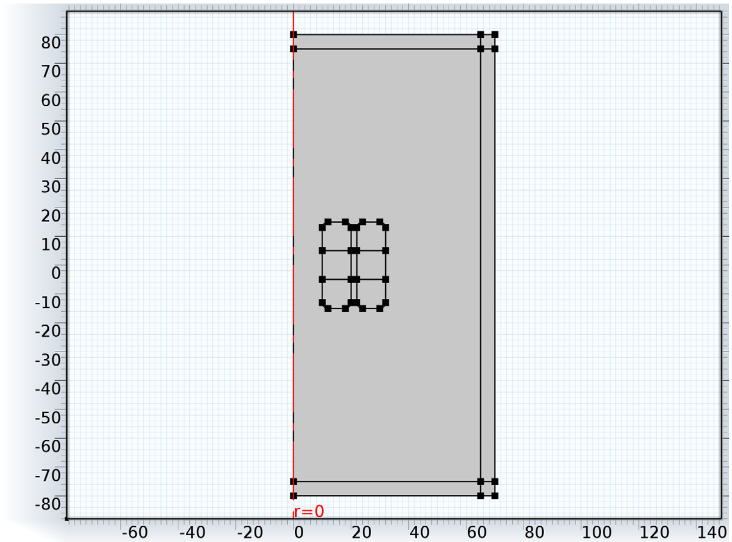
Thickness (mm)
5

7 Select the **Layers to the right** check box.
8 Select the **Layers on top** check box.
9 Click the **Build Selected** button.
10 Click the **Zoom Extents** button on the Graphics toolbar.

Fillet 1

1 Right-click **Geometry 1** and choose **Fillet**.
2 On the object **r2**, select Points 1, 4, 5, and 8 only.
3 On the object **r1**, select Points 1, 4, 5, and 8 only.
4 In the **Fillet** settings window, locate the **Radius** section.
5 In the **Radius** edit field, type 2.

6 Click the **Build All** button.



The model geometry should look like the one shown in the figure above.

DEFINITIONS

Enclose the inner air domain by an Infinite Element Domain to model the surrounding space.

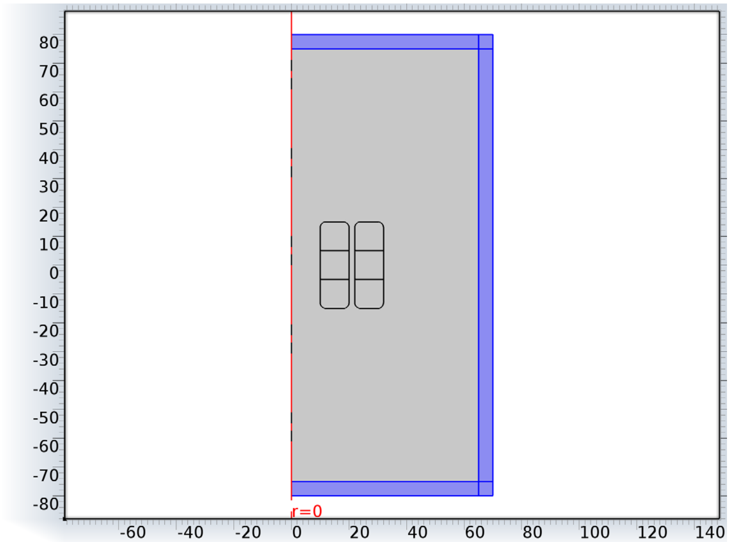
Infinite Element Domain 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Infinite Element Domain**.

2 Select Domains 1, 3, and 10–12 only.

3 In the **Infinite Element Domain** settings window, locate the **Geometry** section.

4 From the **Type** list, choose **Cylindrical**.



MATERIALS

Use air as the material for all domains.

Material Browser

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

2 In the **Material Browser** window, locate the **Materials** section.

3 In the tree, select **Built-In>Air**.

4 Right-click and choose **Add Material to Model** from the menu.

MAGNETIC FIELDS

Now set up the physics for the magnetic field. Use the default Ampère's Law node with default settings for the air domains and add separate nodes (one per magetization direction) for the magnets.

Ampère's Law 2

1 In the **Model Builder** window, under **Model 1** right-click **Magnetic Fields** and choose **Ampère's Law**.

2 Select Domains 6 and 9 only.

3 In the **Ampère's Law** settings window, locate the **Magnetic Field** section.

4 From the **Constitutive relation** list, choose **Remanent flux density**.

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

0	r
0	phi
-Br	z

Ampère's Law 3

1 In the **Model Builder** window, right-click **Magnetic Fields** and choose **Ampère's Law**.

2 Select Domains 4 and 7 only.

3 In the **Ampère's Law** settings window, locate the **Magnetic Field** section.

4 From the **Constitutive relation** list, choose **Remanent flux density**.

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

0	r
0	phi
Br	z

Ampère's Law 4

1 Right-click **Magnetic Fields** and choose **Ampère's Law**.

2 Select Domain 5 only.

3 In the **Ampère's Law** settings window, locate the **Magnetic Field** section.

4 From the **Constitutive relation** list, choose **Remanent flux density**.

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

Br	r
0	phi
0	z

Ampère's Law 5

1 Right-click **Magnetic Fields** and choose **Ampère's Law**.

2 Select Domain 8 only.

3 In the **Ampère's Law** settings window, locate the **Magnetic Field** section.

4 From the **Constitutive relation** list, choose **Remanent flux density**.

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

`-Br`	r
0	phi
0	z

Add a Force Calculation feature to compute the total magnetic force on the inner magnets.

Force Calculation 1

1. Right-click **Magnetic Fields** and choose **Force Calculation**.
2. Select Domains 4–6 only.

   Keeping the default force name, 0, the axial force component can be accessed as `mf.Forcez_0` (‘`mf`’ is the identifier for the Magnetic Fields physics).

Next, add Deformed Geometry and Sensitivity interfaces to use for calculating the axial magnetic stiffness coefficient as defined by Equation 1 of the Model Definition section.

MODEL 1

In the **Model Builder** window, right-click **Model 1** and choose **Add Physics**.

MODEL WIZARD

1. Go to the **Model Wizard** window.
2. In the **Add physics** tree, select **Mathematics>Deformed Mesh>Deformed Geometry (dg)**.
3. Click **Add Selected**.
4. In the **Add physics** tree, select **Mathematics>Optimization and Sensitivity>Sensitivity (sens)**.
5. Click **Add Selected**.
6. Click **Finish**.

DEFORMED GEOMETRY

1. In the **Model Builder** window, under **Model 1** click **Deformed Geometry**.
2. In the **Deformed Geometry** settings window, locate the **Domain Selection** section.
3. Click **Clear Selection**.
4. Click **Paste Selection**.
5. Go to the **Paste Selection** dialog box.

6 In the **Selection** edit field, type `2,4-9`.
7 Click the **OK** button.

Free Deformation 1

1 In the **Model Builder** window, right-click **Deformed Geometry** and choose **Free Deformation**.
2 Select Domains 2 and 4–6 only.

Override the default zero mesh displacement on the outer boundaries of the inner magnets, which are allowed to move in the axial direction.

Prescribed Mesh Displacement 2

1 Right-click **Deformed Geometry** and choose **Prescribed Mesh Displacement**.
2 In the **Prescribed Mesh Displacement** settings window, locate the **Boundary Selection** section.
3 Click **Paste Selection**.
4 Go to the **Paste Selection** dialog box.
5 In the **Selection** edit field, type `8-17,38-41`.
6 Click the **OK** button.
7 In the **Prescribed Mesh Displacement** settings window, locate the **Prescribed Mesh Displacement** section.
8 In the d_z edit field, type `dZ`.

SENSITIVITY

With the Sensitivity interface you can compute the right-hand side of Equation 1 as the derivative of the global objective defined as the axial force component `mf.Forcez_0` with respect to the global control variable defined as the axial displacement `dZ`.

Global Control Variables 1

1 In the **Model Builder** window, under **Model 1** right-click **Sensitivity** and choose **Global Control Variables**.
2 In the **Global Control Variables** settings window, locate the **Control Variables** section.
3 In the **Control variables** table, enter the following settings:

Variable
dZ

Global Objective 1

1 In the **Model Builder** window, right-click **Sensitivity** and choose **Global Objective**.

2 In the **Global Objective** settings window, locate the **Global Objective** section.

3 In the **Objective expression** edit field, type `mf.Forcez_0`.

MESH 1

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

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

3 From the **Element size** list, choose **Finer**.

Use an even finer mesh in the magnet domains.

Size 1

1 Right-click **Model 1>Mesh 1** and choose **Size**.

2 In the **Size** settings window, locate the **Geometric Entity Selection** section.

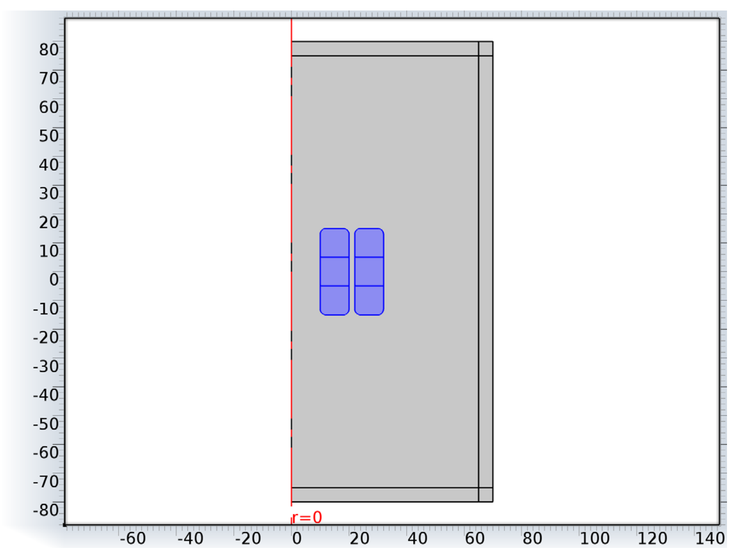
3 From the **Geometric entity level** list, choose **Domain**.

4 Click **Paste Selection**.

5 Go to the **Paste Selection** dialog box.

6 In the **Selection** edit field, type `4-9`.

7 Click the **OK** button.



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

9 From the **Predefined** list, choose **Extremely fine**.

Free Triangular 1

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

2 In the **Free Triangular** settings window, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2 and 4–9 only.

Mapped 1

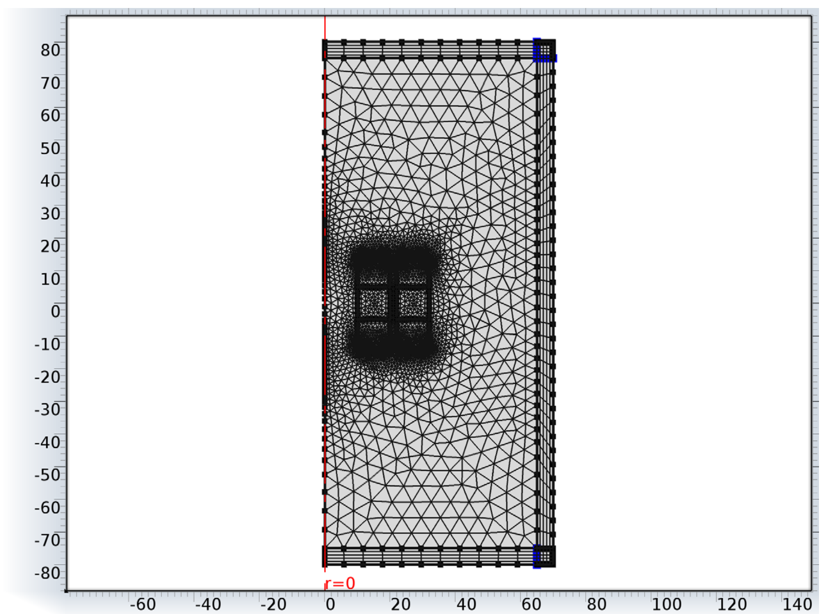
Right-click **Mesh 1** and choose **Mapped**.

Distribution 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 28, 32, and 33 only.

3 Click the **Build All** button.



The mesh should look like that shown in the figure above.

STUDY 1

Add a Parametric Sweep study step to calculate the axial and radial force components for different axial positions of the inner magnets. Vary the axial displacement from z = –40 mm to z = 40 mm.

Parametric Sweep

1 In the **Model Builder** window, right-click **Study 1** and choose **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter names
dZ

5 Click **Range**.

6 Go to the **Range** dialog box.

7 In the **Start** edit field, type `-40`.

8 In the **Step** edit field, type `2`.

9 In the **Stop** edit field, type `40`.

10 Click the **Replace** button.

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

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

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

Solver 1

1 Right-click **Study 1** and choose **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solver 1** node.

3 Right-click **Stationary Solver 1** and choose **Sensitivity**.

4 Right-click **Study 1** and choose **Compute**.

RESULTS

Data Sets

Create data sets for result visualization in specific domains.

1 In the **Model Builder** window, under **Results>Data Sets** right-click **Solution 2** and choose **Duplicate**.

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

3 From the **Solution** list, choose **dZ=8**.

4 Right-click **Results>Data Sets>Solution 3** and choose **Add Selection**.

5 In the **Selection** settings window, locate the **Geometric Entity Selection** section.

6 From the **Geometric entity level** list, choose **Domain**.

7 Select Domains 4–9 only.

8 Select the **Propagate to lower dimensions** check box.

9 In the **Model Builder** window, right-click **Data Sets** and choose **Revolution 2D**.

10 In the **Revolution 2D** settings window, locate the **Data** section.

11 From the **Data set** list, choose **Solution 3**.

12 Locate the **Revolution Layers** section. In the **Start angle** edit field, type `-100`.

13 In the **Revolution angle** edit field, type `280`.

Use the following instructions to get the plots shown in Figure 2 through Figure 6.

2D Plot Group 1

1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.

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

3 Right-click **Results>2D Plot Group 1>Surface 1** and choose **Plot**.

4 Right-click **2D Plot Group 1** and choose **Contour**.

5 In the **Contour** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Magnetic Fields>Magnetic>Magnetic vector potential (Material)>Magnetic vector potential, phi component (Aphi)**.

6 Click the **Plot** button.

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

Compare this plot with Figure 2.

1D Plot Group 2

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

2 In the **1D Plot Group** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Solution 2**.

4 Right-click **Results>1D Plot Group 2** and choose **Global**.

5 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Magnetic**

Fields>Mechanical>Electromagnetic force (Material)>Electromagnetic force, r component (mf.Forcer_0).

6 Click the **Plot** button.

7 In the **Global** settings window, click to expand the **Legends** section.

8 Clear the **Show legends** check box.

The plot should be as shown in Figure 3.

1D Plot Group 3

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

2 In the **1D Plot Group** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Solution 2**.

4 Right-click **Results>1D Plot Group 3** and choose **Global**.

5 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Sensitivity>Objective value (sens.gobj1)**.

6 Locate the **y-Axis Data** section. In the table, enter the following settings:

Description
Fz

7 Click the **Plot** button.

Compare the plot you just generate with that shown in Figure 4.

1D Plot Group 4

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

2 In the **1D Plot Group** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Solution 2**.

4 Right-click **Results>1D Plot Group 4** and choose **Global**.

5 In the **Global** settings window, locate the **y-Axis Data** section.

6 In the table, enter the following description in the 1st row:

Expression	Description
fsens(dZ)	Sensitivity of Fz w.r.t. dZ [N/m]

7 Click on any cell in the 2nd table row and then click the **Delete** button below the table to remove the entry for the objective value.

8 Click the **Plot** button.

Compare this plot with Figure 5.

Finally, reproduce Figure 6 using the following steps.

3D Plot Group 5

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

2 Right-click **3D Plot Group 5** and choose **Volume**.

3 Click the **Plot** button.