Created in COMSOL Multiphysics 5.5

# 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 example illustrates how to calculate design parameters like magnetic forces and stiffness for an axial permanent magnet bearing.

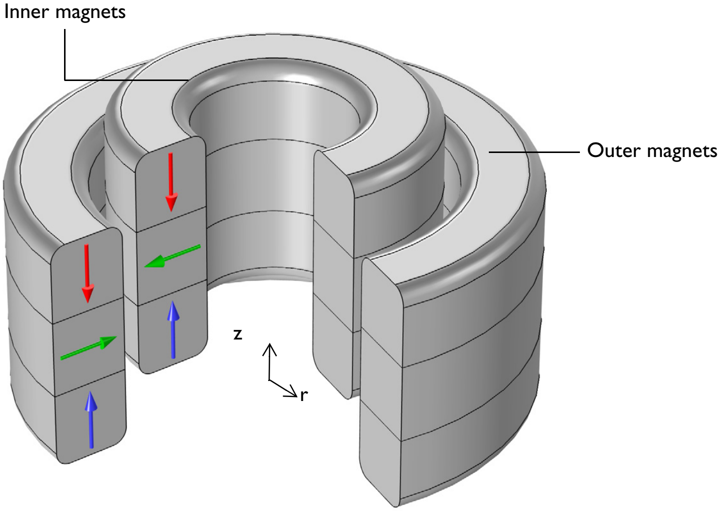


*Figure 1: Model illustration of an axial magnetic bearing using permanent magnets. The black arrows show the magnetization direction of the permanent magnets.*

## 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 $\boldsymbol{F}_z$ 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 illustrates the axial component of the magnetic force on the inner magnets as a function of axial displacement. Figure 4 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 5 shows the magnetic flux density norm in 3D at an axial displacement of 8 mm.

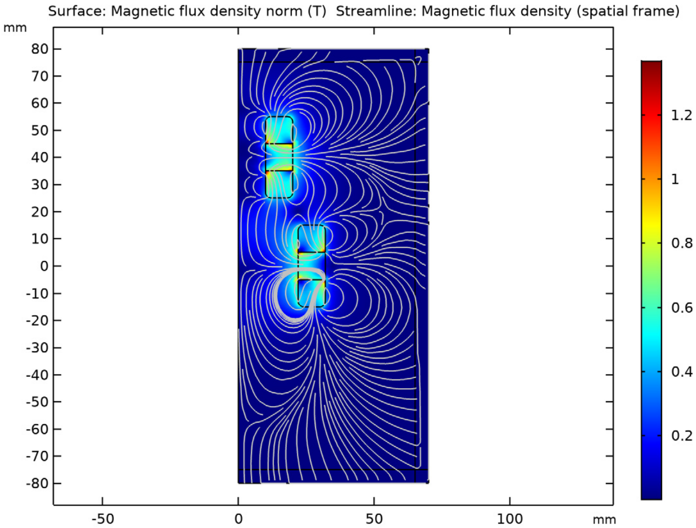


*Figure 2: Magnetic flux density norm and magnetic vector potential for an axial displacement of the inner magnets of 40 mm.*

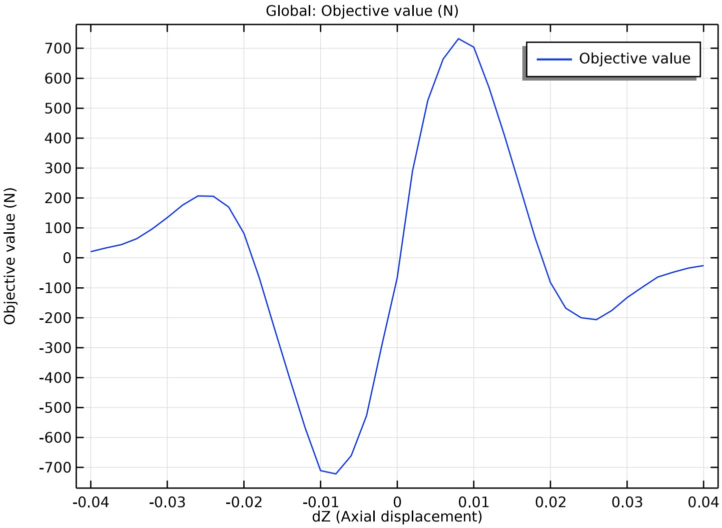


*Figure 3: Axial component of the magnetic force versus axial displacement.*

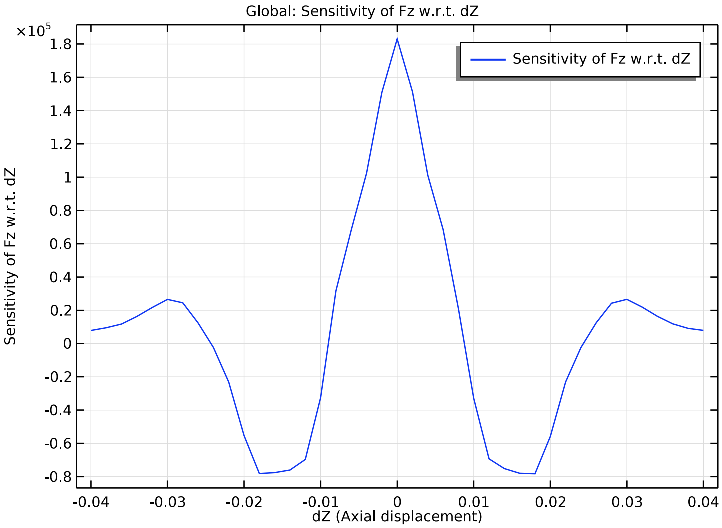


*Figure 4: Sensitivity of the axial magnetic force with respect to axial displacement versus axial displacement. The negative of this quantity is the axial magnetic stiffness coefficient.*

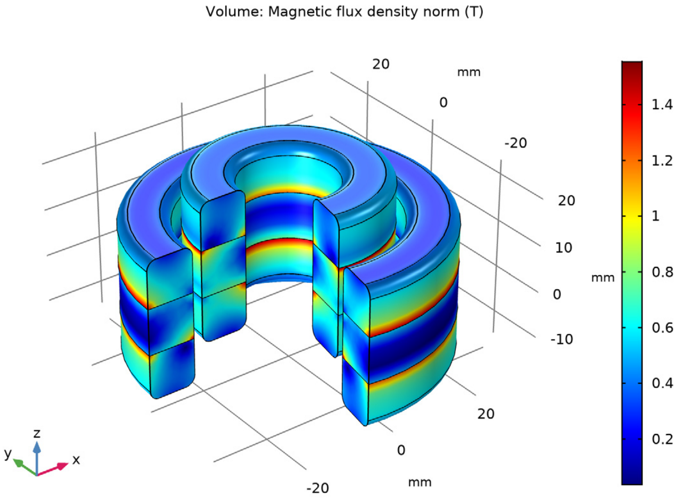


*Figure 5: Magnetic flux density norm at an axial displacement of 8 mm.*

## *Notes About the COMSOL Implementation*

In this model, use the Magnetic Fields interface to model the magnetic field. Also, add an Infinite Element Domain to model the open region of free space surrounding the magnets. You can then 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, 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.

---

**Application Library path:** `ACDC_Module/Motors_and_Actuators/axial_magnetic_bearing`

---

## *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 **2D Axisymmetric**.
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

Define all the necessary parameters here.

## GLOBAL DEFINITIONS

### *Parameters 1*

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

| **Name** | **Expression** | **Value** | **Description** |
|---|---|---|---|
| `R1` | `10[mm]` | 0.01 m | `Inner radius of inner magnet` |
| `R2` | `20[mm]` | 0.02 m | `Outer radius of inner magnet` |
| `R3` | `22[mm]` | 0.022 m | `Inner radius of outer magnet` |
| `R4` | `32[mm]` | 0.032 m | `Outer radius of outer magnet` |
| `h0` | `10[mm]` | 0.01 m | `Magnet height` |
| `Br` | `1[T]` | 1 T | `Remanent flux density of magnet` |
| `dZ` | `0[mm]` | 0 m | `Axial displacement` |

Later, `dZ` will be used as a global control variable for a sensitivity analysis and the parameter of a **Parametric Sweep** node in order to compute the axial magnetic stiffness.

## GEOMETRY 1

Follow the instructions below to construct the model geometry.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

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

### *Rectangle 1 (r1)*

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

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

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

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

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

7 Click **Build Selected**.

8 Click to expand the **Layers** section. In the table, enter the following settings:

| **Layer name** | **Thickness (mm)** |
|---|---|
| `Layer 1` | `h0` |

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

10 Click **Build Selected**.

### *Rectangle 2 (r2)*

1 Right-click **Rectangle 1 (r1)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

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

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

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

6 Click **Build Selected**.

### *Rectangle 3 (r3)*

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `70`.

4 In the **Height** text field, type `160`.

5 Locate the **Position** section. In the **z** text field, type `-80`.

6 Locate the **Layers** section. In the table, enter the following settings:

| **Layer name** | **Thickness (mm)** |
|---|---|
| `Layer 1` | `5` |

7 Select the **Layers to the right** check box.

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

9 Click **Build Selected**.

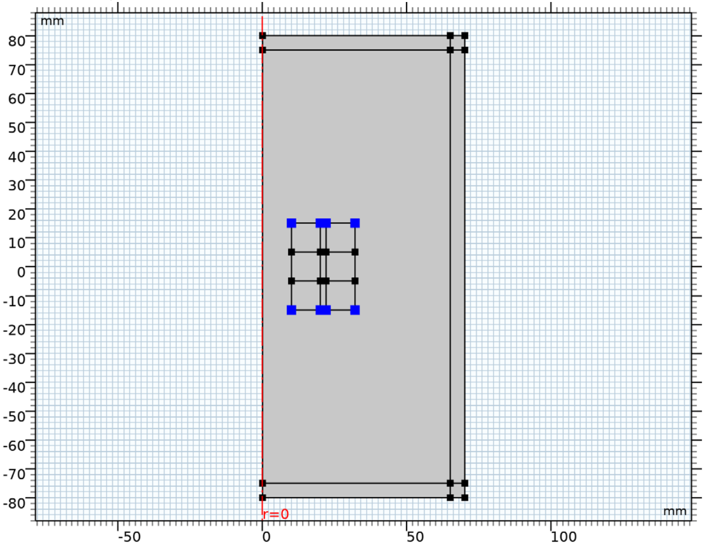
10 Click the **Zoom Extents** button in the **Graphics** toolbar.

### *Fillet 1 (fil1)*

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

2 On the object **r1**, select Points 1, 4, 5, and 8 only.

**3** On the object **r2**, select Points 1, 4, 5, and 8 only.

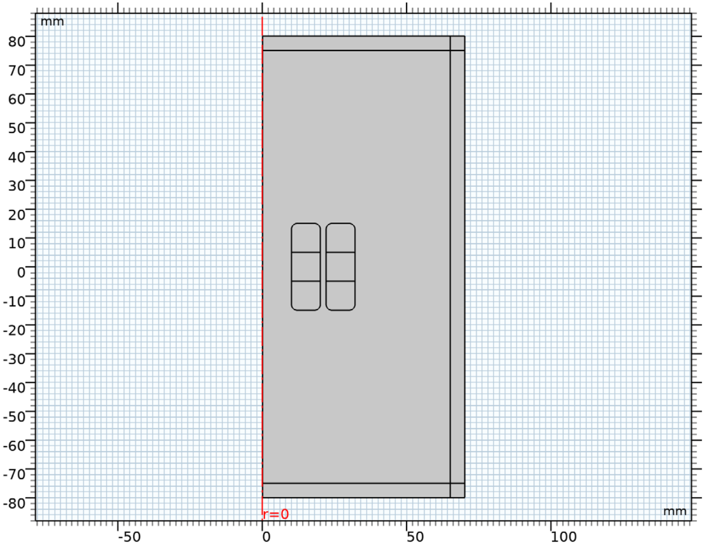


**4** In the **Settings** window for **Fillet**, locate the **Radius** section.

**5** In the **Radius** text field, type `2`.

**6** Click **Build All Objects**.

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

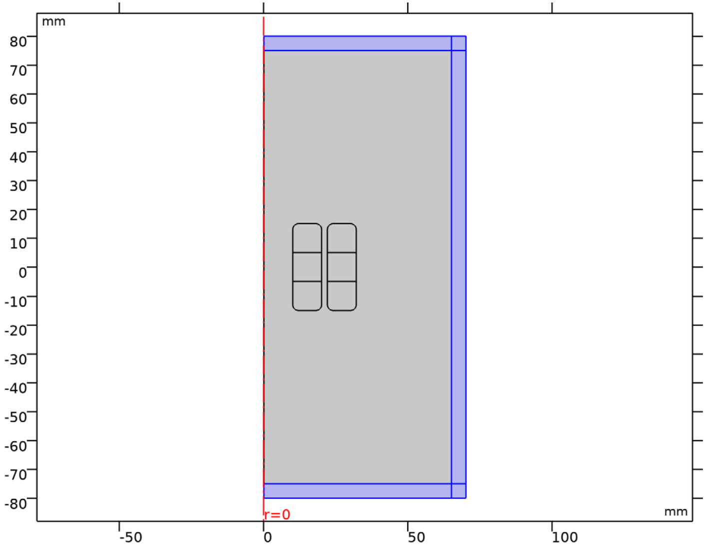


The model geometry should look like the one shown in the figure above.

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

### *Infinite Element Domain 1 (ie1)*

1. In the **Definitions** toolbar, click **Infinite Element Domain**.
2. Select Domains 1, 3, and 10–12 only.
3. In the **Settings** window for **Infinite Element Domain**, locate the **Geometry** section.
4. From the **Type** list, choose **Cylindrical**.



## MATERIALS

Use air as the material for all domains and override it in permanent magnets with a material that will be created based on a modification of an existing entry from library.

1. In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

## ADD MATERIAL

1. Go to the **Add Material** window.
2. In the tree, select **Built-in>Air**.
3. Click **Add to Component** in the window toolbar.
4. In the tree, select **AC/DC>Hard Magnetic Materials> Sintered NdFeB Grades (Chinese Standard)>N50 (Sintered NdFeB)**.
5. Click **Add to Component 1 (comp1)**.

6 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

### MATERIALS

*N50 (Sintered NdFeB) (mat2)*

1 Select Domains 4–9 only.

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

3 In the table, enter the following settings:

| **Property** | **Variable** | **Value** | **Unit** | **Property group** |
|---|---|---|---|---|
| Recoil permeability | murec_iso ; murecii = murec_iso, murecij = 0 | 1 | 1 | Remanent flux density |
| Remanent flux density norm | normBr | `Br` | T | Remanent flux density |

4 In the **Label** text field, type `Generic magnet`.

Now set up the physics for the magnetic field: use the default **Magnetic Flux Conservation** node with default settings for the air domains and add separate nodes for the magnets (one per magnetization direction).

### 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 Domains 6 and 9 only.

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

4 From the **Magnetization model** list, choose **Remanent flux density**.

5 Specify the **e** vector as

| | |
|---|---|
| 0 | r |
| 0 | phi |
| -1 | z |

### *Magnetic Flux Conservation 3*

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

2 Select Domains 4 and 7 only.

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

4 From the **Magnetization model** list, choose **Remanent flux density**.

5 Specify the **e** vector as

| | |
|---|---|
| 0 | r |
| 0 | phi |
| 1 | z |

### *Magnetic Flux Conservation 4*

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

2 Select Domain 5 only.

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

4 From the **Magnetization model** list, choose **Remanent flux density**.

### *Magnetic Flux Conservation 5*

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

2 Select Domain 8 only.

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

4 From the **Magnetization model** list, choose **Remanent flux density**.

5 Specify the **e** vector as

| | |
|---|---|
| -1 | r |
| 0 | phi |
| 0 | z |

### *Force Calculation 1*

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

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

2 Select Domains 4–6 only.

Keeping the default force name, `0`, the axial force component can be accessed as `mfnc.Forcez_0`, where `mfnc` is the identifier for the **Magnetic Fields, No Currents** interfaces.

Specify a reference level for the magnetic scalar potential by constraining its value in one point.

### *Zero Magnetic Scalar Potential 1*

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

2 Select Point 30 only.

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

## ADD PHYSICS

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

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

3 In the tree, select **Mathematics>Optimization and Sensitivity>Sensitivity (sens)**.

4 Click **Add to Component 1** in the window toolbar.

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

## SENSITIVITY (SENS)

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 `mfnc.Forcez_0` with respect to the global control variable defined as the axial displacement `dZ`.

### *Global Control Variables 1*

1 Right-click **Component 1 (comp1)>Sensitivity (sens)** and choose **Global Control Variables**.

2 In the **Settings** window for **Global Control Variables**, locate the **Control Variables** section.

3 In the **Control variables** table, enter the following settings:

| Variable | Initial value |
|---|---|
| dZ | 0 |

### *Global Objective 1*

1 In the **Physics** toolbar, click **Global** and choose **Global Objective**.

2 In the **Settings** window for **Global Objective**, locate the **Global Objective** section.

3 In the *q* text field, type `mfnc.Forcez_0`.

## DEFINITIONS

### *Deforming Domain 1*

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Deforming Domain**.

2 Select Domain 2 only.

### *Prescribed Deformation 1*

In the **Definitions** toolbar, click **Moving Mesh** and choose **Prescribed Deformation**.

### *Deforming Domain 1*

1 In the **Model Builder** window, click **Deforming Domain 1**.

2 In the **Settings** window for **Deforming Domain**, locate the **Smoothing** section.

3 From the **Mesh smoothing type** list, choose **Laplace**.

### *Prescribed Deformation 1*

1 Click the **Select All** button in the **Graphics** toolbar.

2 In the **Model Builder** window, click **Prescribed Deformation 1**.

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

4 Click **Clear Selection**.

5 Click the **Select Box** button in the **Graphics** toolbar.

6 Select Domains 4–6 only.

7 Locate the **Prescribed Deformation** section. Specify the *dx* vector as

| | |
|---|---|
| 0 | R |
| dZ | Z |

### *Prescribed Mesh Displacement 1*

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Prescribed Mesh Displacement**.

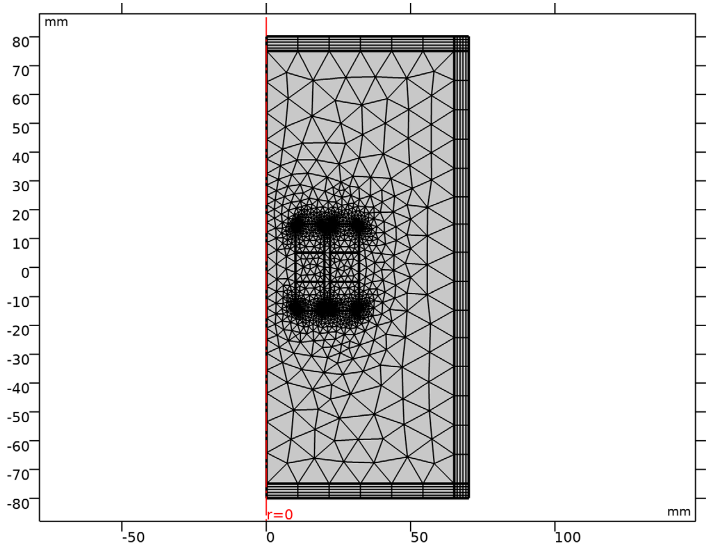
2 Select Boundaries 3, 4, 6, 18, 19, 21, 23–27, 30, and 42–45 only.

## MESH 1

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

2 In the **Settings** window for **Mesh**, click **Build All**.

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



The mesh should look like the one shown in the figure above. Notice that the physics controlled meshing automatically produced a mapped mesh in the infinite elements.

### 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 **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 From the list in the **Parameter name** column choose **dZ (Axial displacement)**, and set the **Parameter unit** to mm.

5 Click **Range**.

6 In the **Range** dialog box, type `-40` in the **Start** text field.

7 In the **Step** text field, type `2`.

8 In the **Stop** text field, type `40`.

9 Click **Replace**.

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

11 In the **Settings** window for **Study**, locate the **Study Settings** section.

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

### *Solution 1 (sol1)*

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

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

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

## RESULTS

In the **Model Builder** window, expand the **Results** node.

### *Study 1/Parametric Solutions 1 (sol2)*

Create data sets for result visualization in specific domains.

### *Study 1/Parametric Solutions 1 (3) (sol2)*

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets>Study 1/Parametric Solutions 1 (sol2)** and choose **Duplicate**.

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

4 From the **Solution** list, choose **dZ=8 (sol27)**.

### *Selection*

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Select Domains 4–9 only.

### *Revolution 2D 1*

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/dZ=8 (sol27)**.

4 Click to expand the **Revolution Layers** section. In the **Start angle** text field, type `-100`.

5 In the **Revolution angle** text field, type `280`.

Use the following instructions to get the plots shown in Figure 2 to Figure 5.

### *2D Plot Group 1*

In the **Results** toolbar, click **2D Plot Group**.

### *Surface 1*

1. Right-click **2D Plot Group 1** and choose **Surface**.
2. In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields, No Currents>Magnetic>mfnc.normB - Magnetic flux density norm - T**.
3. In the **2D Plot Group 1** toolbar, click **Plot**.

### *Streamline 1*

1. In the **Model Builder** window, right-click **2D Plot Group 1** and choose **Streamline**.
2. In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
3. From the **Positioning** list, choose **Uniform density**.
4. In the **Separating distance** text field, type `0.02`.
5. Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **Gray**.
6. In the **2D Plot Group 1** toolbar, click **Plot**.
7. Click the **Zoom Extents** button in the **Graphics** toolbar.

   Compare this plot with Figure 2.

### *1D Plot Group 2*

In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

### *Global 1*

1. Right-click **1D Plot Group 2** and choose **Global**.
2. In the **Settings** window for **Global**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.
4. Locate the **y-Axis Data** section. Click **Clear Table**.
5. Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Sensitivity>sens.gobj1 - Objective value - N**.
6. In the **1D Plot Group 2** toolbar, click **Plot**.

   Compare the plot just created with that shown in Figure 3.

### *1D Plot Group 3*

In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

### *Global 1*

1. Right-click **1D Plot Group 3** and choose **Global**.
2. In the **Settings** window for **Global**, locate the **Data** section.

3. From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.
4. Locate the **y-Axis Data** section. Click **Clear Table**.
5. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| `fsens(dZ)` | `N/m` | `Sensitivity of Fz w.r.t. dZ` |

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

   Compare this plot with Figure 4.

### *3D Plot Group 4*

Finally, reproduce Figure 5 using the following steps.

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

### *Volume 1*

1. Right-click **3D Plot Group 4** and choose **Volume**.
2. In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **mfnc.normB - Magnetic flux density norm - T**.
3. In the **3D Plot Group 4** toolbar, click **Plot**.