Model created in COMSOL Multiphysics 6.4

Axial Magnetic Bearing Using Permanent Magnets

Introduction

Permanent magnet bearings are used in turbomachinery, 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.

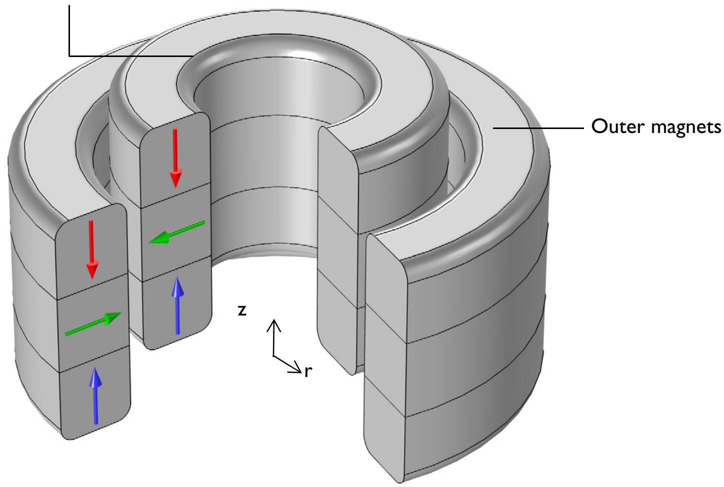


Figure 1: Model illustration of an axial magnetic bearing using permanent magnets. The black arrows show the magnetization direction of the permanent magnets.

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 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.

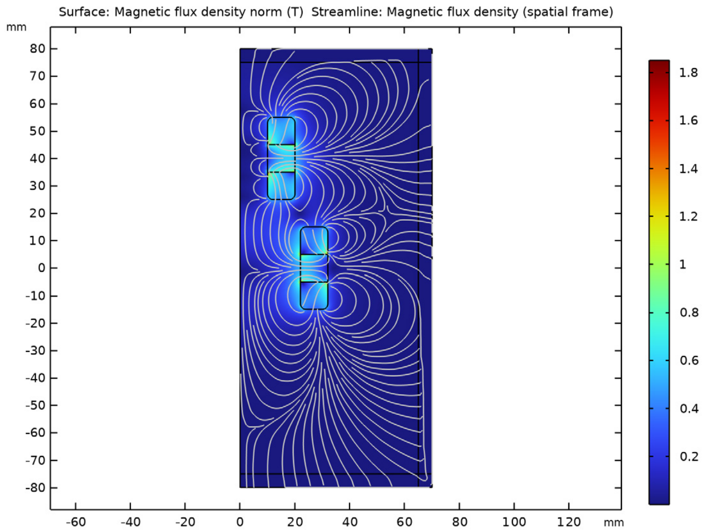


Figure 2: Magnetic flux density norm and magnetic vector potential for an axial displacement of the inner magnets of 40 mm.

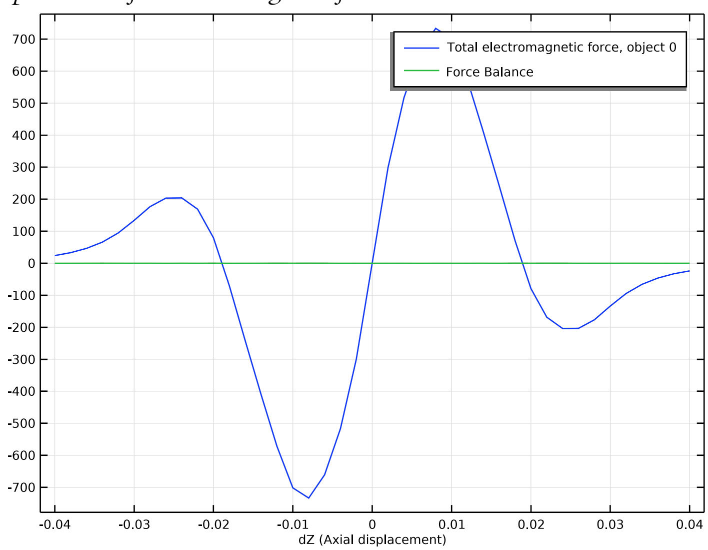


Figure 3: Axial component of the magnetic force versus axial displacement.

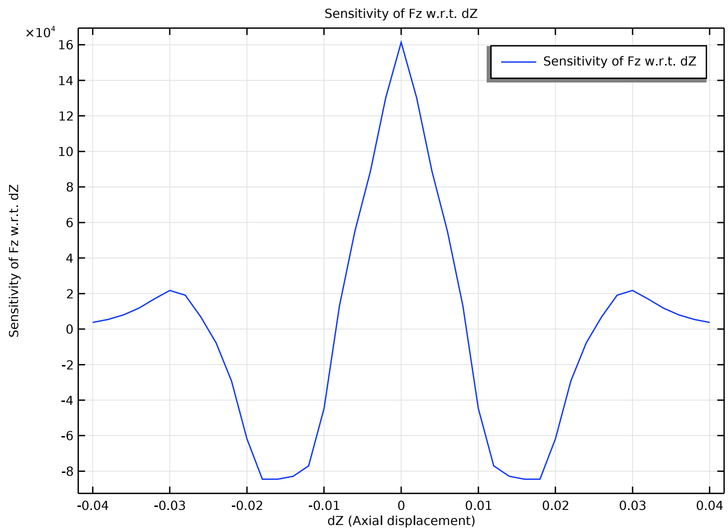


Figure 4: Sensitivity of the axial magnetic force with respect to axial displacement versus axial displacement. The negative of this quantity is the axial magnetic stiffness coefficient.

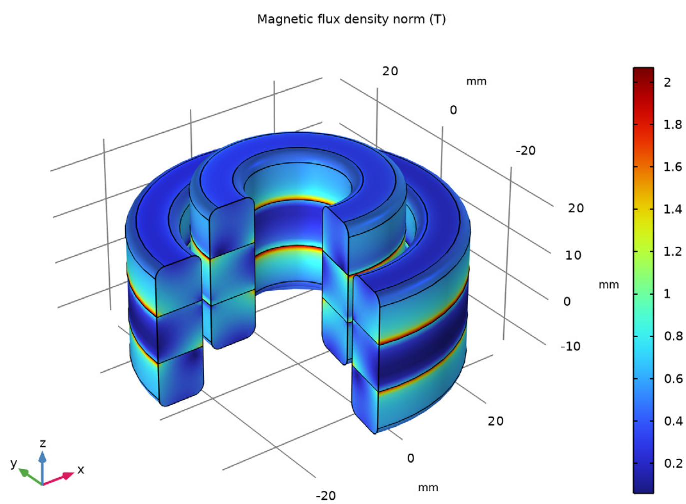


Figure 5: Magnetic flux density norm at an axial displacement of 8 mm.

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 Moving Mesh features and a Sensitivity interface to calculate the axial magnetic stiffness coefficient as defined by Equation 1. With the Moving Mesh features 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, G. Lemarquand and V. Lemarquand, “Halbach Structures for Permanent Magnets Bearings,” *Progress In Electromagnetics Research M*, vol. 14, pp. 236–277, 2010.

Application Library path: `ACDC_Module/Devices,_Transducers_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** checkbox.

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** checkbox.

8 Select the **Layers on top** checkbox.

9 Click **Build Selected**.

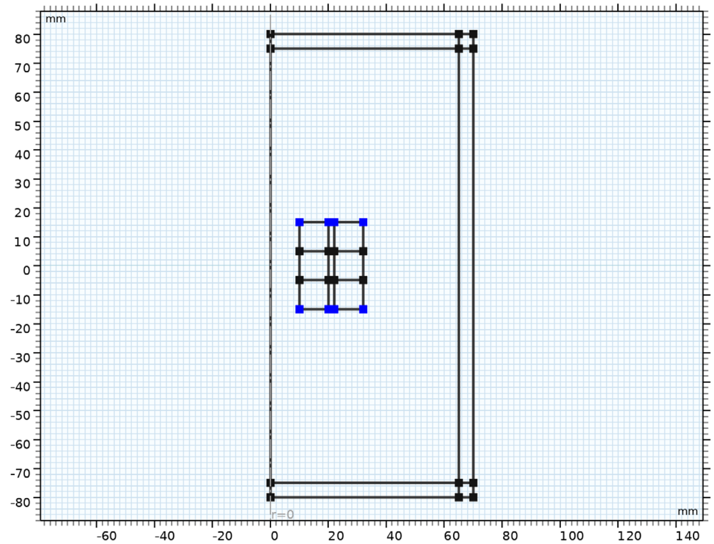
10 Click the **Zoom Extents** button in the **Graphics** toolbar.

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.

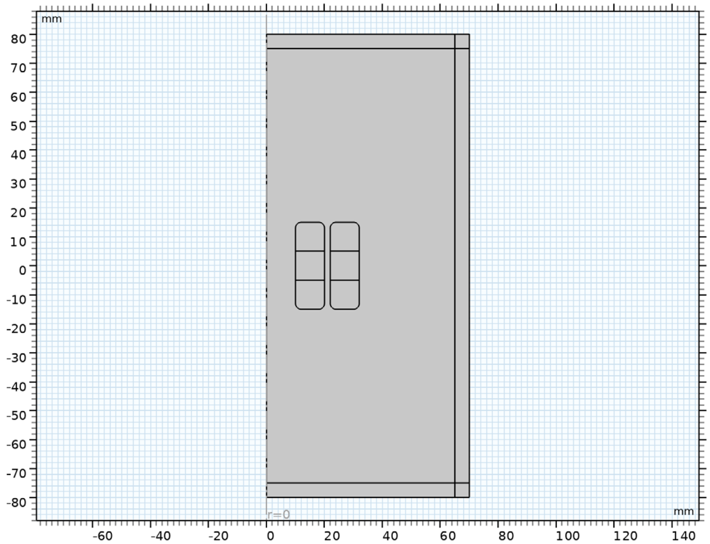


4 In the **Settings** window for **Fillet**, locate the **Radius** section.

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.

DEFINITIONS

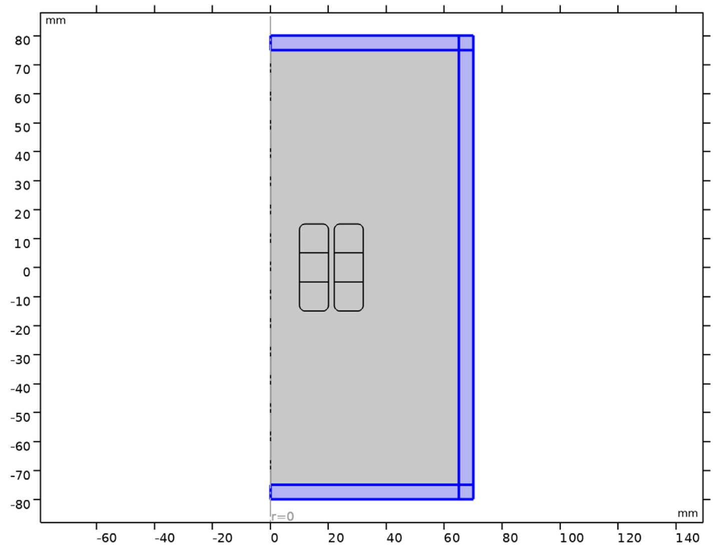
Infinite Element Domain 1 (ie1)

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**.



Now set up the physics for the magnetic field: use the **Magnetic Flux Conservation in Solids** node for the magnets (one per magnetization direction).

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 6 and 9 only.

3 In the **Settings** window for **Magnetic Flux Conservation in Solids**, 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 in Solids 2

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

2 Select Domains 4 and 7 only.

3 In the **Settings** window for **Magnetic Flux Conservation in Solids**, 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 in Solids 3

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

2 Select Domain 5 only.

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

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

Magnetic Flux Conservation in Solids 4

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

2 Select Domain 8 only.

3 In the **Settings** window for **Magnetic Flux Conservation in Solids**, 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

MATERIALS

In the magnets, use a material that will be created by modifying an existing entry from the library.

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)** > **N50 (Sintered NdFeB)**.

4 Right-click and choose **Add to Component 1 (comp1)**.

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

MATERIALS

Generic Magnet

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`.

MAGNETIC FIELDS, NO CURRENTS (MFNC)

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

Force Calculation 1

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

2 Select Domains 4–9 only.

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

4 In the **Force name** text field, type `0`.

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

By using the split by connectivity the inner and the out magnets sets are separated in different features. Outer is automatically identified with name `1`. Labels are changed according to this naming.

Force Calculation, Object 1

1 Right-click **Force Calculation 1** and choose **Split by Connectivity**.

2 In the **Settings** window for **Force Calculation**, type `Force Calculation, Object 1` in the **Label** text field.

Force Calculation, Object 0

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Magnetic Fields, No Currents (mfnc)** click **Force Calculation 1**.

2 In the **Settings** window for **Force Calculation**, type `Force Calculation, Object 0` in the **Label** text field.

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 the **Add to Component 1** button 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 In the **Physics** toolbar, click **Global** 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 (1)
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`.

COMPONENT 1 (COMP1)

Deforming Domain 1

1 In the **Physics** toolbar, click **Moving Mesh** and choose **Free Deformation**.

2 Select Domain 2 only.

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

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

Prescribed Deformation 1

1 In the **Moving Mesh** toolbar, click **Prescribed Deformation**.

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

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

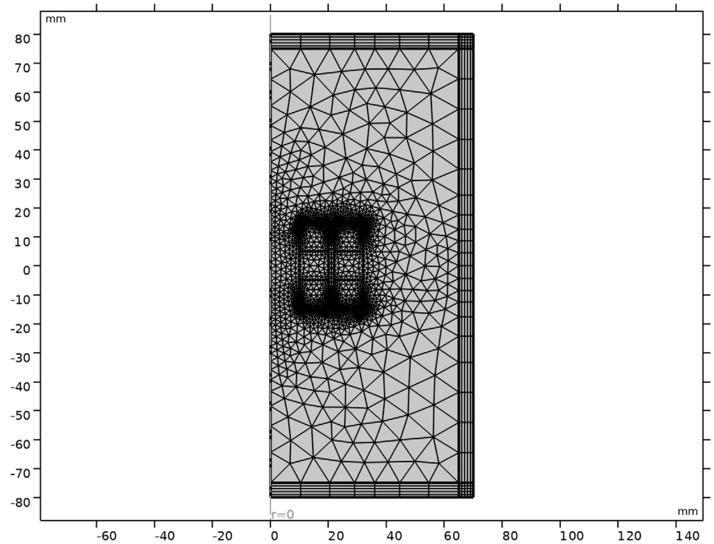
1 In the **Moving Mesh** toolbar, click **Prescribed Mesh Displacement**.

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)** right-click **Mesh 1** and choose **Build All**.

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



The mesh should look like the one shown in the figure above. Note 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, 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** checkbox.

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 **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **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 datasets 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.

Magnetic Flux Density, 2D Visualization

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

2 In the **Settings** window for **2D Plot Group**, type `Magnetic Flux Density, 2D Visualization` in the **Label** text field.

Surface 1

1 Right-click **Magnetic Flux Density, 2D Visualization** 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 (comp1)** > **Magnetic Fields, No Currents** > **Magnetic** > **mfnc.normB - Magnetic flux density norm - T**.

3 In the **Magnetic Flux Density, 2D Visualization** toolbar, click **Plot**.

Streamline 1

1 In the **Model Builder** window, right-click **Magnetic Flux Density, 2D Visualization** 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 **Density level** text field, type `7.77`.

5 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **Gray**.

6 In the **Magnetic Flux Density, 2D Visualization** toolbar, click **Plot**.

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

Compare this plot with Figure 2.

Force vs dZ and Force Balance

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Force vs dZ and Force Balance` in the **Label** text field.

Global 1

1 Right-click **Force vs dZ and Force Balance** 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 (comp1)** > **Sensitivity** > **sens.gobj1 - Objective value - N**.

Add to this plot also the force balance.

6 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Magnetic Fields, No Currents** > **Mechanical** > **Total electromagnetic force, object 0 (spatial frame) - N** > **mfnc.Forcez_0 - Total electromagnetic force, object 0, component z**.

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

Expression	Unit	Description
`sens.gobj1`	`N`	`Total electromagnetic force, object 0`
`mfnc.Forcez_0+ mfnc.Forcez_1`	`N`	`Force Balance`

8 In the **Force vs dZ and Force Balance** toolbar, click **Plot**.

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

9 Click **Plot**.

Magnetic Stiffness Coefficient

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Magnetic Stiffness Coefficient` in the **Label** text field.

Global 1

1 Right-click **Magnetic Stiffness Coefficient** 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 **Magnetic Stiffness Coefficient** toolbar, click **Plot**.

Compare this plot with Figure 4.

Magnetic Fields, Revolution Visualization

Finally, reproduce Figure 5 using the following steps.

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

2 In the **Settings** window for **3D Plot Group**, type `Magnetic Fields, Revolution Visualization` in the **Label** text field.

Volume 1

1 Right-click **Magnetic Fields, Revolution Visualization** 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 **Magnetic Fields, Revolution Visualization** toolbar, click **Plot**.