



Model created in COMSOL Multiphysics 6.4

# Magnetic–Structure Interaction in a Permanent Magnet Motor

## *Introduction*

---

Permanent Magnet (PM) motors are widely used in a variety of domestic and industrial applications including electric vehicles, high speed railways, aerospace and HVAC applications. Though high in initial cost, PM motors provide high efficiency over a large operational speed and power range, which make them suitable for many of these robust machineries.

In this example, the coupling between the Multibody Dynamics interface and the Rotating Machinery, Magnetic interface for performing mechanical and electromagnetic analysis is demonstrated. A permanent magnet motor with surface mounted magnets is modeled in 2D. To model magnetic–structure coupling integrated with moving mesh, the electromagnetic force is transferred to the rotor, and the rotor motion is transferred to the moving mesh. A time-dependent problem, computing the magnetic flux density and displacement, is solved for three electrical periods.

---

**Note:** This model requires the AC/DC Module and the Multibody Dynamics Module.

---

## *Model Definition*

---

The main parts of an electric motor are a moving rotor, housed in a stationary stator, separated by an air gap to enable the rotation. In a PM motor, magnets are attached either to the surface of the rotor or embedded inside them.

In this example, a PM motor with 10 rotor poles and 12 stator slots is modeled in 2D. The diameter of rotor and stator are 150 mm 250 mm respectively. The axial length of the motor is 300 mm. As shown in [Figure 1](#), the magnets are mounted on the surface of the rotor. The interaction between the magnetic field of the rotor and the magnetic field generated by stator currents produces the driving torque. To reduce the weight and minimize material usage, there are five air channels in the rotor.

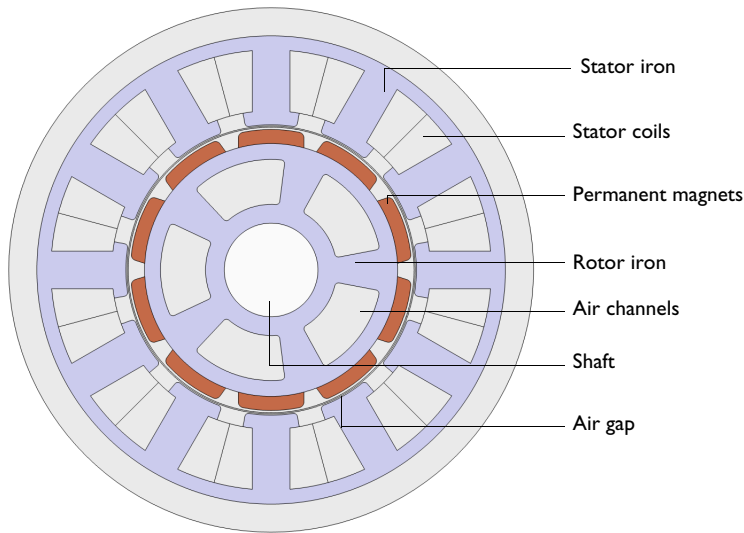


Figure 1: Geometry of the motor.

### MAGNETIC-STRUCTURE INTERACTION

The interaction between electromagnetic and structural domains is modeled using a Magnetic-Rigid Body Interaction in Rotating Machinery interface. This interface consists of a Rotating Machinery, Magnetic interface, a Multibody Dynamics interface, and a **Moving Mesh** node with a **Deforming Domain** and a **Rotating Boundary** subnodes. In addition, a **Multiphysics Couplings** node is added. It contains the multiphysics coupling **Magnetic Forces, Rotating Machinery**. Using this functionality, the electromagnetic forces generated during the rotation of the motor is transferred to the structural domains.

Additional details about the interface can be found in the documentation for Multiphysics Couplings in the *Multibody Dynamics Module User's Guide*.

### ROTATING MACHINERY, MAGNETIC

Rotating Machinery, Magnetic interface is used to solve the electromagnetic field equations in a transverse section of the PM motor. The stator and rotor iron are made of silicon steel with zero conductivity. The permanent magnets are made of NdFeB, creating a strong magnetic field. The center shaft is made of high strength alloy steel. The rotational speed is taken as 700 rpm. The stator coil is excited with a peak current of 10 A, with an initial current angle for peak torque set as 198°.

## MULTIBODY DYNAMICS

The Multibody Dynamics interface is used to model the rotor and permanent magnets. For mechanical analysis, they are considered as elastic with properties as shown in [Table 1](#).

TABLE 1: MECHANICAL PROPERTIES OF ROTOR IRON AND MAGNETS.

Property	Symbol	Unit	Silicon steel	NdFeB
Young's modulus	$E$	GPa	195	160
Poisson's ratio	$\nu$	l	0.25	0.24
Mass density	$\rho$	kg/m <sup>3</sup>	7700	7600

The effect of centrifugal force generated by the rotation of the rotor is modeled using a **Rotating Frame** node.

## MOVING MESH

A deforming domain condition is assigned to the rotor air gap and other rotor air channel domains, which experience significant deformation due to the rotation of adjacent structural domains. The shape of these domains is controlled by the moving boundaries and a smoothing equation in the interior. On the external boundaries of the rotor air gap, a **Rotating Boundary** condition is used to enable the sliding of the mesh.

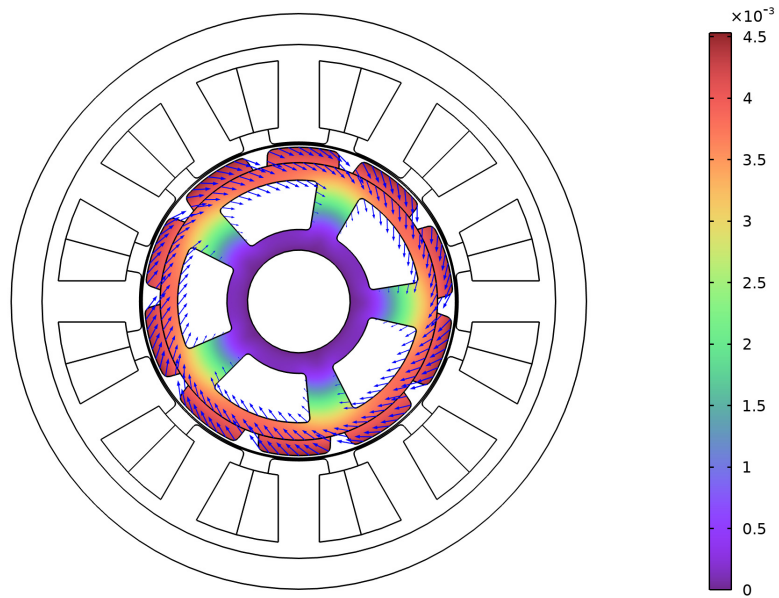
## *Results and Discussion*

---

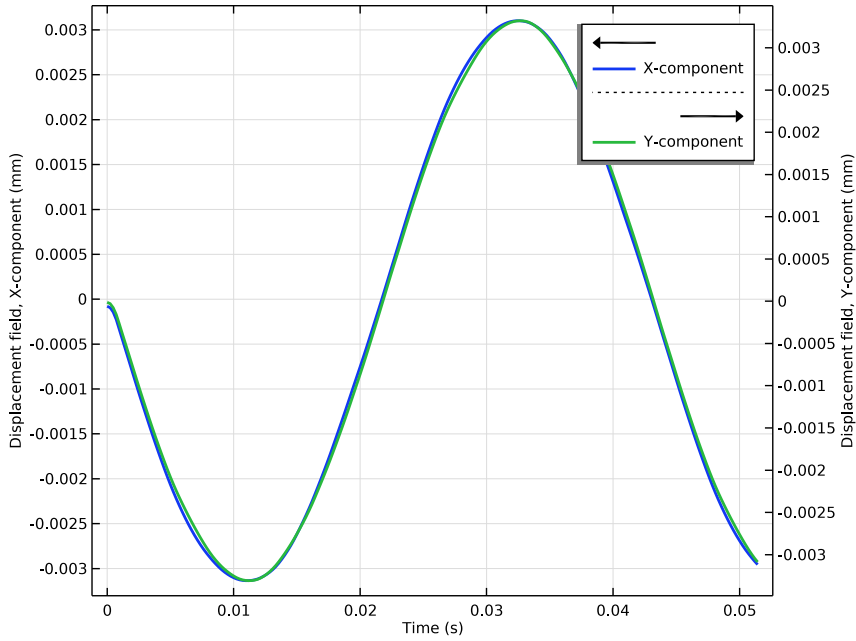
A time-dependent problem is solved for three electrical periods.

[Figure 2](#) displays the total displacement of rotor, with arrows showing the direction of displacement at the end of three electrical periods. In [Figure 3](#), the displacement of a sample point on the rotor core is plotted as a function of time.

Time=0.051429 s Surface: Displacement magnitude (mm) Arrow Line: Displacement field

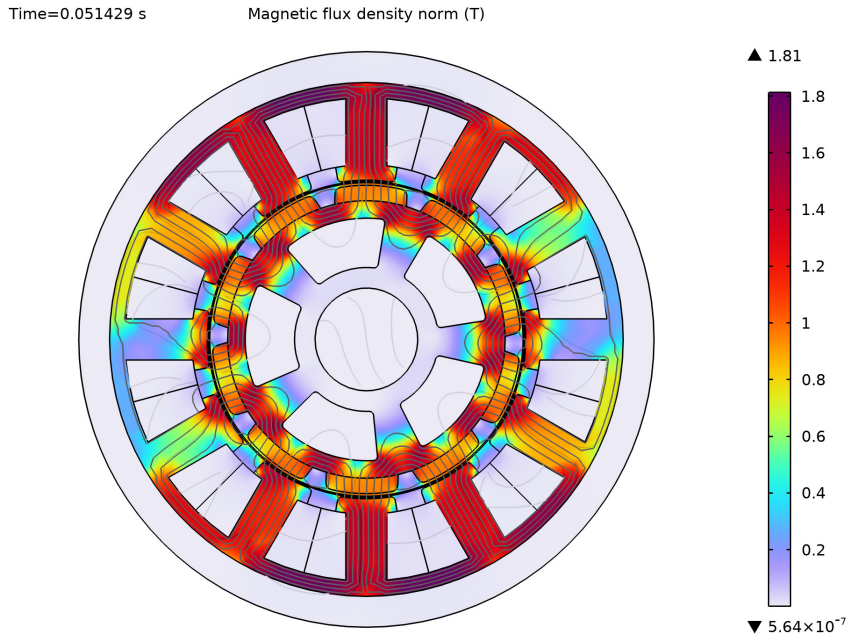


*Figure 2: Displacement of rotor at the end of three electrical periods. The arrows show the direction of displacements relative to a corotating frame.*

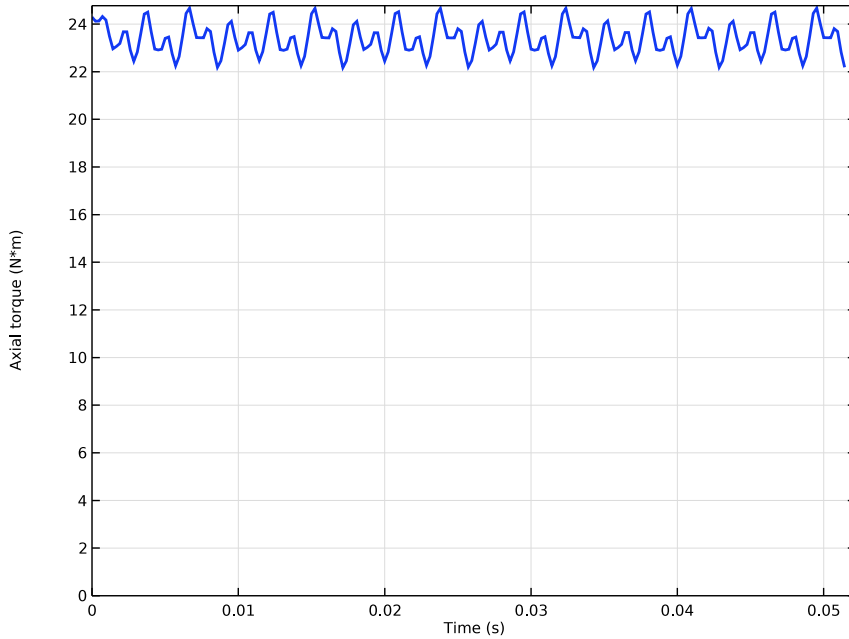


*Figure 3: Displacement of a sample point on rotor iron as a function of time for three electrical periods.*

Figure 4 and Figure 5 show plots from the electromagnetic analysis. In Figure 4, the norm of the magnetic flux density and field lines are shown. Figure 5 plots the rotor torque ripple as a function of time for three electrical periods.



*Figure 4: The norm and field lines of magnetic flux density at the end of three electrical periods.*



*Figure 5: Rotor torque plotted as a function of time for three complete electrical periods.*

---

### *Notes About the COMSOL Implementation*

In order to get appropriate initial conditions for the time-dependent analysis, a stationary solution is run first. This will establish a state of initial deformations and strains, caused by the magnetic field and centrifugal forces.

---


**Application Library path:** ACDC\_Module/Devices,\_Motors\_and\_Generators/pm\_motor\_2d\_structure\_interaction

---




### *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**.
- 2 In the **Select Physics** tree, select **Structural Mechanics > Electromagnetics–Structure Interaction > Magnetomechanics > Rotating Machinery, Magnetic–Structure Interaction > Magnetic–Rigid-Body Interaction in Rotating Machinery**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies > Stationary**.
- 6 Click  **Done**.

## GEOMETRY I


Change the units to mm.

- 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**.

Begin by specifying a number of general parameters that will be used in the model.



## 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 Click  **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `pm_motor_2d_structure_interaction_parameters.txt`.

Next, build the motor using rotor and stator parts from the geometry part library. Initialize the parts, and tick the selections that are predefined to make it convenient to assign material properties and magnetization direction.

## PART LIBRARIES


- 1 In the **Home** toolbar, click  **Windows** and choose **Part Libraries**.
- 2 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
- 3 In the **Part Libraries** window, select **AC/DC Module > Rotating Machinery 2D > Rotors > Internal > surface\_mounted\_magnet\_internal\_rotor\_2d** in the tree.
- 4 Click  **Add to Geometry**.

## GEOMETRY I


### *Internal Rotor – Surface Mounted Magnets I (pi1)*

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Geometry 1** click **Internal Rotor – Surface Mounted Magnets I (pi1)**.
- 2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
number_of_poles	Np	10	Number of magnetic poles in rotor
number_of_modeled_poles	Np	10	Number of magnetic poles included in the geometry
magnet_h	1.5 [mm]	1.5 mm	Height of the magnets
magnet_w	7 [mm]	7 mm	Width of the magnets (set to 0 to use all available space)


- 4 Click to expand the **Domain Selections** section. In the table, select the **Keep** checkboxes for **Shaft**, **Rotor Magnets**, and **Rotor air**.
- 5 Click  **Build Selected**.

### *Circle 1 (c1)*


- 1 In the **Geometry** toolbar, click  **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 11.8.
- 4 In the **Sector angle** text field, type 360/APnr\*APfct.
- 5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	4.8


### *Extract 1 (extract1)*

- 1 In the **Geometry** toolbar, click  **Extract**.
- 2 In the **Settings** window for **Extract**, locate the **Entities or Objects to Extract** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 On the object **c1**, select Domain 2 only.
- 5 From the **Input object handling** list, choose **Remove**.


#### *Fillet 1 (fill)*

- 1 In the **Geometry** toolbar, click  **Fillet**.
- 2 On the object **extract1**, select Points 1–4 only.
- 3 In the **Settings** window for **Fillet**, locate the **Radius** section.
- 4 In the **Radius** text field, type  $2/APnr$ .


#### *Rotate: Rotor Air Channels*

- 1 In the **Geometry** toolbar, click  **Transforms** and choose **Rotate**.
- 2 Select the object **fill** only.
- 3 In the **Settings** window for **Rotate**, type Rotate: Rotor Air Channels in the **Label** text field.
- 4 Locate the **Rotation** section. In the **Angle** text field, type  $\text{range}((360/APnr - 360/APnr * APfct) / 2, 360/APnr, 360)$ .
- 5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.


#### *Rotating Boundaries*

- 1 In the **Geometry** toolbar, click  **Selections** and choose **Disk Selection**.
- 2 In the **Settings** window for **Disk Selection**, type Rotating Boundaries in the **Label** text field.
- 3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
- 4 Locate the **Size and Shape** section. In the **Outer radius** text field, type  $\text{inf}$ .
- 5 In the **Inner radius** text field, type  $30.5/2 * 0.99$ .

#### *Extract 2 (extract2)*




- 1 In the **Geometry** toolbar, click  **Extract**.
- 2 In the **Settings** window for **Extract**, locate the **Entities or Objects to Extract** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 On the object **pi1**, select Domain 12 only.
- 5 From the **Input object handling** list, choose **Create remainder object**.

#### *Difference: Rotor Core*

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.
- 2 In the **Settings** window for **Difference**, type Difference: Rotor Core in the **Label** text field.
- 3 Select the object **extract2(1)** only.

- 4 Locate the **Difference** section. From the **Objects to subtract** list, choose **Rotate: Rotor Air Channels**.
- 5 Select the **Keep objects to subtract** checkbox.
- 6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

*Union: Rotor*

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Union**.
- 2 In the **Settings** window for **Union**, type Union: Rotor in the **Label** text field.
- 3 Click the  **Select All** button in the **Graphics** toolbar.
- 4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.
- 5 Click  **Build Selected**.



*Circle 1 (c1), Difference: Rotor Core (dif1), Extract 1 (extract1), Extract 2 (extract2), Fillet 1 (fil1), Internal Rotor – Surface Mounted Magnets 1 (pi1), Rotate: Rotor Air Channels (rot1), Rotating Boundaries (diskse1), Union: Rotor (uni1)*

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Geometry 1**, Ctrl-click to select **Internal Rotor – Surface Mounted Magnets 1 (pi1)**, **Circle 1 (c1)**, **Extract 1 (extract1)**, **Fillet 1 (fil1)**, **Rotate: Rotor Air Channels (rot1)**, **Rotating Boundaries (diskse1)**, **Extract 2 (extract2)**, **Difference: Rotor Core (dif1)**, and **Union: Rotor (uni1)**.
- 2 Right-click and choose **Group**.

*Stator*

In the **Settings** window for **Group**, type Stator in the **Label** text field.

## PART LIBRARIES

- 1 In the **Geometry** toolbar, click  **Part Libraries**.
- 2 In the **Model Builder** window, click **Geometry 1**.
- 3 In the **Part Libraries** window, select **AC/DC Module > Rotating Machinery 2D > Stators > External > slotted\_external\_stator\_2d** in the tree.
- 4 Click  **Add to Geometry**.


## GEOMETRY 1

*External Stator – Slotted 1 (pi2)*


Specify number of slots and select a radial partition for the slot winding type.

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Geometry 1** click **External Stator – Slotted 1 (pi2)**.
- 2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
number_of_slots	Ns	12	Number of slots in stator
number_of_modeled_slots	Ns	12	Number of slots included in the geometry
slot_winding_type	2	2	Slot winding type: 1-No partition, 2-Radial partition, 3-Azimuthal partition, 4-Radial and azimuthal partition.
Arkio_toggle	1	1	Toggle Arkio air gap - (1/0) (on/off)



- 4 Locate the **Domain Selections** section. In the table, select the **Keep** checkboxes for **Stator iron**, **Stator slots**, and **All domains**.
- 5 Click  **Build Selected**.

#### *Scale 1 (scal)*

- 1 In the **Geometry** toolbar, click  **Transforms** and choose **Scale**.
- 2 Click in the **Graphics** window and then press Ctrl+A to select both objects.
- 3 In the **Settings** window for **Scale**, locate the **Scale Factor** section.
- 4 In the **Factor** text field, type geom\_scale.


Create an assembly from the two geometry objects, connected by a pair boundary.

#### *Form Union (fin)*

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Geometry 1** click **Form Union (fin)**.
- 2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.
- 3 From the **Action** list, choose **Form an assembly**.
- 4 In the **Geometry** toolbar, click  **Build All**.
- 5 Click the  **Zoom Extents** button in the **Graphics** toolbar.



## DEFINITIONS

### *Ramp 1 (rm1)*




- 1 In the **Definitions** toolbar, click  **More Functions** and choose **Ramp**.
- 2 In the **Settings** window for **Ramp**, locate the **Parameters** section.
- 3 In the **Slope** text field, type 1/t\_ramp.
- 4 Select the **Cutoff** checkbox.

Next, add selections to assign materials and physics features.



### *Structural Domains*

- 1 In the **Definitions** toolbar, click  **Union**.
- 2 In the **Settings** window for **Union**, type Structural Domains in the **Label** text field.
- 3 Locate the **Input Entities** section. Under **Selections to add**, click  **Add**.
- 4 In the **Add** dialog, in the **Selections to add** list, choose **Rotor Magnets (Internal Rotor – Surface Mounted Magnets 1)** and **Difference: Rotor Core**.
- 5 Click **OK**.

### *Deforming Domains*



- 1 Right-click **Structural Domains** and choose **Duplicate**.
- 2 In the **Settings** window for **Union**, type Deforming Domains in the **Label** text field.
- 3 Locate the **Input Entities** section. In the **Selections to add** list box, select **Rotor Magnets (Internal Rotor – Surface Mounted Magnets 1)**.
- 4 Under **Selections to add**, click  **Delete**.
- 5 Under **Selections to add**, click  **Delete**.
- 6 Under **Selections to add**, click  **Add**.
- 7 In the **Add** dialog, in the **Selections to add** list, choose **Rotor air (Internal Rotor – Surface Mounted Magnets 1)** and **Rotate: Rotor Air Channels**.
- 8 Click **OK**.

### *Shaft Boundaries*

- 1 In the **Definitions** toolbar, click  **Adjacent**.
- 2 In the **Settings** window for **Adjacent**, type Shaft Boundaries in the **Label** text field.
- 3 Locate the **Input Entities** section. Under **Input selections**, click  **Add**.
- 4 In the **Add** dialog, select **Shaft (Internal Rotor – Surface Mounted Magnets 1)** in the **Input selections** list.
- 5 Click **OK**.

Next, add materials and assign them to their appropriate domain selections.

#### 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 **Built-in > Air**.
- 4 Click the **Add to Component** button in the window toolbar.
- 5 In the tree, select **Nonlinear Magnetic > Silicon Steel NGO > Silicon Steel NGO 35PN440**.
- 6 Click the **Add to Component** button in the window toolbar.
- 7 In the tree, select **AC/DC > Magnetic Materials (Bomatec®) > NdFeB Sintered > BMN-42**.
- 8 Click the **Add to Component** button in the window toolbar.
- 9 In the tree, select **Built-in > High-strength alloy steel**.
- 10 Click the **Add to Component** button in the window toolbar.
- 11 In the **Materials** toolbar, click  **Add Material** to close the **Add Material** window.

#### MATERIALS

*Silicon Steel NGO 35PN440 (mat2)*

- 1 Select Domains 2 and 35 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
Young's modulus	E	195 [GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.25	I	Young's modulus and Poisson's ratio
Density	rho	7700	kg/m <sup>3</sup>	Basic
Electric conductivity	sigma_iso ; sigma <sub>ii</sub> = sigma_iso, sigma <sub>ij</sub> = 0	0	S/m	Basic

#### *BMN-42 (mat3)*

- 1 In the **Model Builder** window, click **BMN-42 (mat3)**.
- 2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Rotor Magnets (Internal Rotor – Surface Mounted Magnets 1)**.
- 4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	160 [GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.24		Young's modulus and Poisson's ratio
Density	rho	7600	kg/m <sup>3</sup>	Basic

#### *High-strength alloy steel (mat4)*

- 1 In the **Model Builder** window, click **High-strength alloy steel (mat4)**.
- 2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Shaft (Internal Rotor – Surface Mounted Magnets 1)**.

#### **MULTIBODY DYNAMICS (MBD)**

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Multibody Dynamics (mbd)**.
- 2 In the **Settings** window for **Multibody Dynamics**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Structural Domains**.
- 4 Locate the **Thickness** section. In the *d* text field, type L.

#### *Rotating Frame 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Multibody Dynamics (mbd)** click **Rotating Frame 1**.
- 2 In the **Settings** window for **Rotating Frame**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Structural Domains**.
- 4 Locate the **Rotating Frame** section. From the **Rotational velocity** list, choose **Revolutions per time**.
- 5 In the  $\Omega_r$  text field, type  $w\_rot*rm1(t[1/s])$ .

#### *Fixed Constraint 1*


- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Fixed Constraint**.

- 2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Shaft Boundaries**.


### **ROTATING MACHINERY, MAGNETIC (RMM)**

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Rotating Machinery, Magnetic (rmm)**.
- 2 In the **Settings** window for **Rotating Machinery, Magnetic**, locate the **Thickness** section.
- 3 In the  $d$  text field, type L.

#### *Ampère's Law 2*

- 1 In the **Physics** toolbar, click  **Domains** and choose **Ampère's Law**.
- 2 Select Domains 2 and 35 only.
- 3 In the **Settings** window for **Ampère's Law**, locate the **Constitutive Relation B-H** section.
- 4 From the **Magnetization model** list, choose **B-H curve**.

#### *Conducting Magnet 1*

- 1 In the **Physics** toolbar, click  **Domains** and choose **Conducting Magnet**.
- 2 In the **Settings** window for **Conducting Magnet**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Rotor Magnets (Internal Rotor – Surface Mounted Magnets 1)**.
- 4 Locate the **Magnet** section. From the **Pattern type** list, choose **Circular pattern**.
- 5 From the **Type of periodicity** list, choose **Alternating**.

#### *North 1*


- 1 In the **Model Builder** window, click **North 1**.
- 2 Select Boundary 314 only.

#### *South 1*


- 1 In the **Model Builder** window, click **South 1**.
- 2 Select Boundary 311 only.

The **Multiphase Winding** feature simplifies excitation of stator coils of electrical machines. For three-phase systems, an automatic ordering of coil domains into a balanced stator winding is supported, provided that the electrical machine topology in terms of number of poles and slots can accommodate it. In the following steps, use a **Multiphase Winding** feature to automatically populate the selections of three subnodes with coil domains representing each phase.

### *Multiphase Winding 1*

- 1 In the **Physics** toolbar, click  **Domains** and choose **Multiphase Winding**.
- 2 In the **Settings** window for **Multiphase Winding**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Stator slots (External Stator – Slotted 1)**.
- 4 Locate the **Multiphase Winding** section. In the  $I_{pk}$  text field, type  $I_{pk}$ .
- 5 In the  $\alpha_i$  text field, type  $init\_ang$ .
- 6 In the  $f_t$  text field, type  $f\_e1*rm1(t[1/s])$ .
- 7 From the **Winding layout configuration** list, choose **Automatic three phase**.
- 8 In the  $n_{poles}$  text field, type  $Np$ .
- 9 In the  $n_{slots}$  text field, type  $Ns$ .
- 10 In the **Number of coils per slot** text field, type 2.
- 11 Locate the **Homogenized Conductor** section. From the list, choose **Filling factor**.
- 12 Locate the **Multiphase Winding** section. Click **Add Phases**.

### *Arkkio Torque Calculation 1*

- 1 In the **Physics** toolbar, click  **Domains** and choose **Arkkio Torque Calculation**.
- 2 Select Domains 1–28 and 30–46 only.

## **MOVING MESH**

### *Deforming Domain 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Moving Mesh** click **Deforming Domain 1**.
- 2 In the **Settings** window for **Deforming Domain**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Deforming Domains**.

### *Rotating Boundary 1*


- 1 In the **Model Builder** window, click **Rotating Boundary 1**.
- 2 In the **Settings** window for **Rotating Boundary**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Rotating Boundaries**.
- 4 Locate the **Rotation** section. From the **Rotation type** list, choose **Specified rotational velocity**.
- 5 From the **Rotational velocity expression** list, choose **General revolutions per time**.
- 6 In the  $f$  text field, type  $w\_rot*rm1(t[1/s])$ .

Adjust the default mesh to ensure sufficient resolution.


## MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Edit Physics-Induced Sequence**.

### Size


- 1 In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** click **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 10.
- 5 In the **Minimum element size** text field, type 1.
- 6 In the **Curvature factor** text field, type 0.5.
- 7 Click  **Build All**.

### Size 1


- 1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 29 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section.
- 7 Select the **Maximum element size** checkbox. In the associated text field, type 1.3.
- 8 Click  **Build All**.


## STUDY 1

### Step 2: Time Dependent



- 1 In the **Study** toolbar, click  **Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 3 In the **Output times** text field, type range(0, t\_step, t\_end).

### Solution 1 (sol1)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

- 3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- 4 From the **Method** list, choose **BDF**.
- 5 Find the **Algebraic variable settings** subsection. In the **Fraction of initial step for Backward Euler** text field, type 0.01.
- 6 From the **Error estimation** list, choose **Exclude algebraic**.
- 7 In the **Model Builder** window, expand the **Study 1 > Solver Configurations > Solution 1 (sol1) > Time-Dependent Solver 1 > Segregated 1** node, then click **Magnetic Potential**.
- 8 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.
- 9 From the **Nonlinear method** list, choose **Automatic (Newton)**.
- 10 In the **Study** toolbar, click  **Compute**.

## RESULTS

- 1 Click the  **Show Grid** button in the **Graphics** toolbar.
- 2 Click the  **Zoom Extents** button in the **Graphics** toolbar.


Follow the instructions below to plot the system displacement as shown in [Figure 2](#).

### *Arrow Line 1*

- 1 In the **Model Builder** window, right-click **Displacement (mbd)** and choose **Arrow Line**.
- 2 In the **Settings** window for **Arrow Line**, locate the **Arrow Positioning** section.
- 3 In the **Number of arrows** text field, type 1500.
- 4 Locate the **Coloring and Style** section.
- 5 Select the **Scale factor** checkbox. In the associated text field, type 2000.
- 6 From the **Color** list, choose **Blue**.


Follow the instructions below to plot the displacement of a sample point on rotor as shown in [Figure 3](#).

### *Graph Plot Style 1*

- 1 In the **Results** toolbar, click  **Configurations** and choose **Graph Plot Style**.
- 2 In the **Settings** window for **Graph Plot Style**, locate the **Coloring and Style** section.
- 3 Find the **Line style** subsection. From the **Color** list, choose **Cycle**.
- 4 Locate the **Legends** section. Find the **Include in automatic mode** subsection. Select the **Description** checkbox.

- 5 Clear the **Solution** checkbox.
- 6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Width** list, choose **2**.

#### *Displacement*

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **Displacement** in the **Label** text field.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 4 Click to expand the **Style Configuration** section. From the **Configuration** list, choose **Graph Plot Style 1**.

#### *Point Graph 1*

- 1 Right-click **Displacement** and choose **Point Graph**.
- 2 Select **Point 222** only.
- 3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 4 In the **Expression** text field, type **u**.
- 5 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 6 From the **Legends** list, choose **Manual**.
- 7 In the table, enter the following settings:

<b>Legends</b>
X-component

#### *Point Graph 2*

- 1 Right-click **Point Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type **v**.
- 4 Locate the **Legends** section. In the table, enter the following settings:

<b>Legends</b>
Y-component


#### *Displacement*

- 1 In the **Model Builder** window, click **Displacement**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Plot Settings** section.
- 3 Select the **Two y-axes** checkbox.
- 4 In the table, select the **Plot on secondary y-axis** checkbox for **Point Graph 2**.

- 5 Select the **x-axis label** checkbox.
- 6 Select the **y-axis label** checkbox.
- 7 Select the **Secondary y-axis label** checkbox.

Follow the instructions below to plot the axial torque as shown in [Figure 5](#).

#### *Torque*

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Torque in the **Label** text field.
- 3 Locate the **Title** section. From the **Title type** list, choose **None**.
- 4 Locate the **Style Configuration** section. From the **Configuration** list, choose **Graph Plot Style 1**.
- 5 Locate the **Legend** section. Clear the **Show legends** checkbox.


#### *Global 1*

- 1 Right-click **Torque** and choose **Global**.
- 2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
rmm.Tark_1	N*m	Axial torque

- 4 In the **Torque** toolbar, click  **Plot**.

#### *Torque*

- 1 In the **Model Builder** window, click **Torque**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Axis** section.
- 3 Select the **Manual axis limits** checkbox.
- 4 In the **x minimum** text field, type 0.
- 5 In the **y minimum** text field, type 0.
- 6 In the **Torque** toolbar, click  **Plot**.

#### *Displacement, Displacement (mbd), Velocity (mbd)*

- 1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Displacement (mbd)**, **Velocity (mbd)**, and **Displacement**.
- 2 Right-click and choose **Group**.

#### *Structural Plots*

In the **Settings** window for **Group**, type Structural Plots in the **Label** text field.


### *Magnetic Flux Density (rmm), Torque*

- 1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Magnetic Flux Density (rmm)** and **Torque**.
- 2 Right-click and choose **Group**.

### *Electromagnetic Plots*

In the **Settings** window for **Group**, type Electromagnetic Plots in the **Label** text field.

### *Displacement (mbd)*

- 1 In the **Results** toolbar, click  **Animation** and choose **Player**.
- 2 In the **Settings** window for **Animation**, type Displacement (mbd) in the **Label** text field.
- 3 Locate the **Frames** section. In the **Number of frames** text field, type 50.

### *Magnetic Flux Density Norm (rmm)*

- 1 Right-click **Displacement (mbd)** and choose **Duplicate**.
- 2 In the **Settings** window for **Animation**, type Magnetic Flux Density Norm (rmm) in the **Label** text field.
- 3 Locate the **Scene** section. From the **Subject** list, choose **Magnetic Flux Density (rmm)**.