Created in COMSOL Multiphysics 6.2

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.

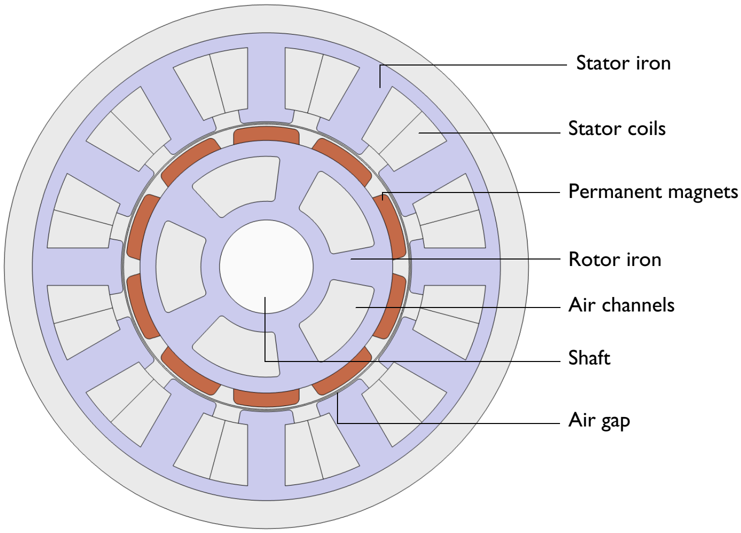

Figure 1: Geometry of the motor.

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	ν	1	0.25	0.24
Mass density	ρ	kg/m^3	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.

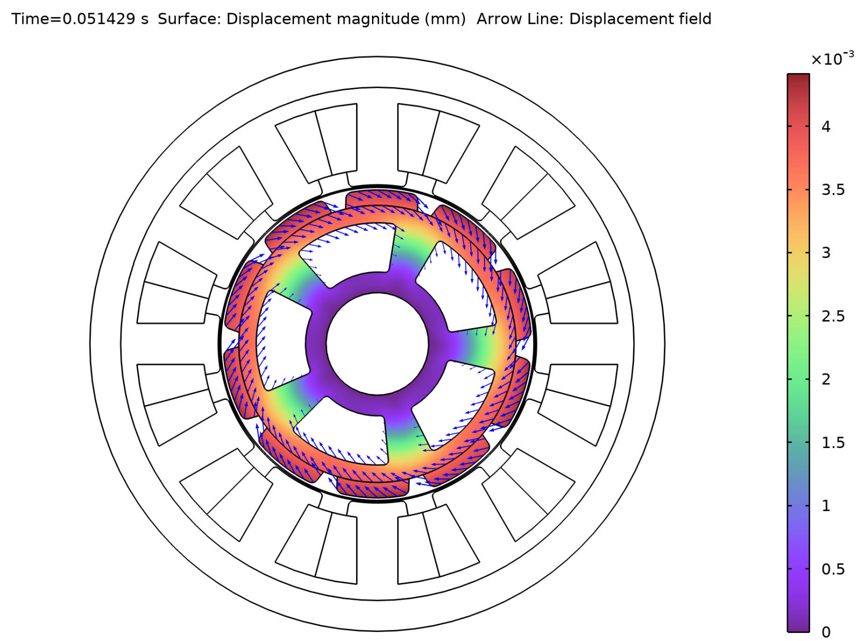

Figure 2: Displacement of rotor at the end of three electrical periods. The arrows show the direction of displacements relative to a corotating frame.

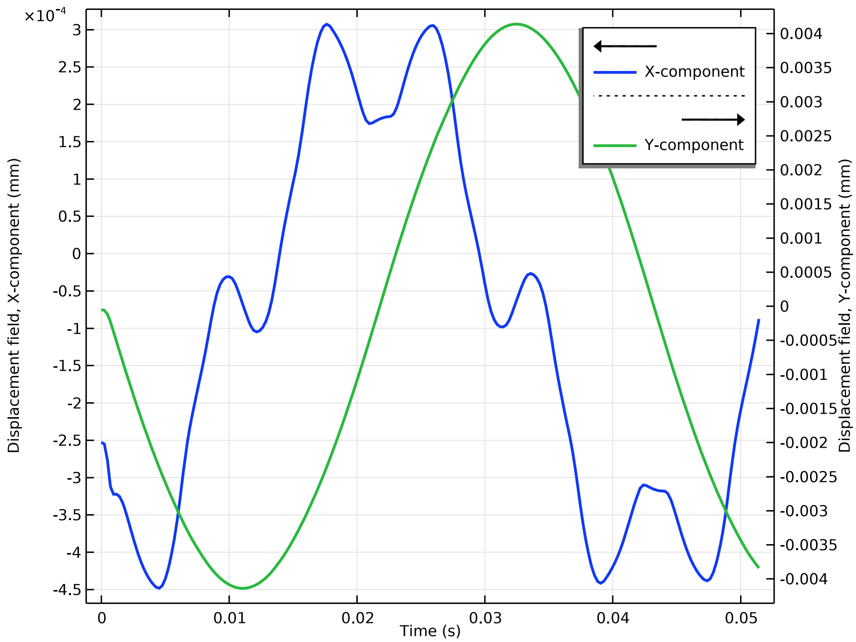

Figure 3: Displacement of a sample point on rotor iron as a function of time for three electrical periods.

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.

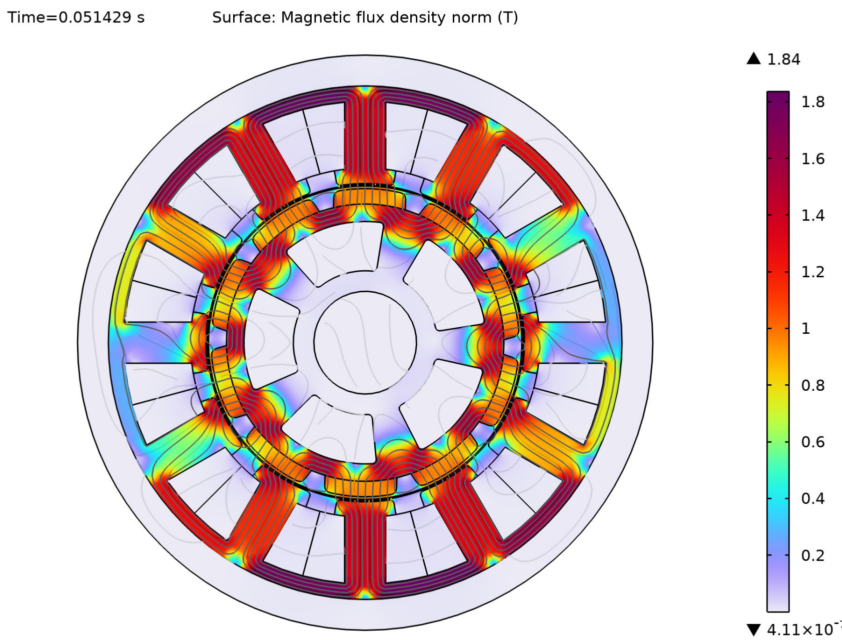

Figure 4: The norm and field lines of magnetic flux density at the end of three electrical periods.

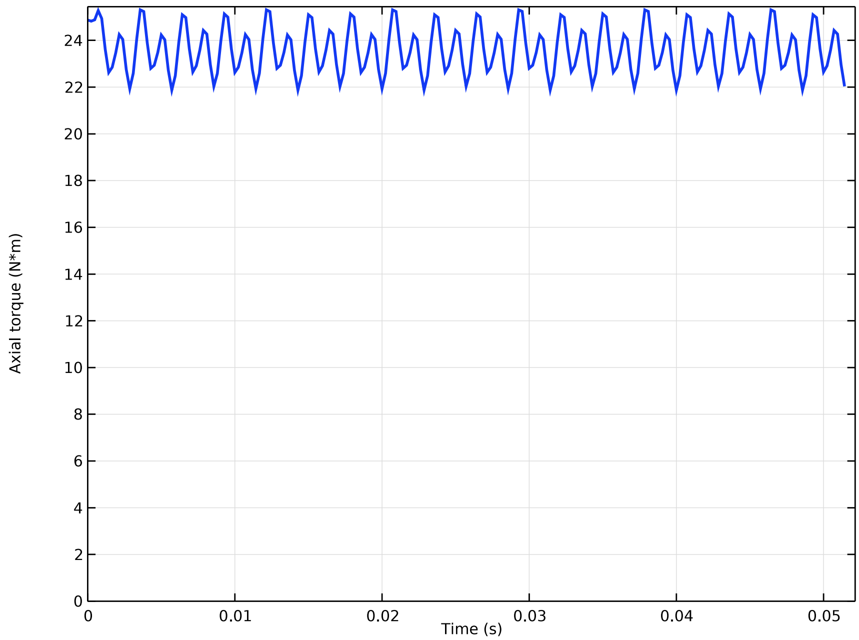

Figure 5: Rotor torque plotted as a function of time for three complete electrical periods.

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: `Multibody_Dynamics_Module/Electrical Machinery/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 1

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 1

Internal Rotor – Surface Mounted Magnets 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Internal Rotor – Surface Mounted Magnets 1 (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** check boxes for **Shaft**, **Rotor Magnets**, and **Rotor air**.
5 Click **Build Selected**.

Split 1 (spl1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Split**.
2 Select the object **pi1** only.
3 In the **Settings** window for **Split**, 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 **Build Selected**.
6 Right-click **Circle 1 (c1)** and choose **Duplicate**.

Circle 2 (c2)

1 In the **Model Builder** window, click **Circle 2 (c2)**.
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3 In the **Radius** text field, type `7`.

4 Click **Build Selected**.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **c1** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **c2** only.

6 Click **Build Selected**.

Fillet 1 (fil1)

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

2 On the object **dif1**, 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`.

5 Click **Build Selected**.

Rotate: Rotor Air Channels

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 In the **Settings** window for **Rotate**, type `Rotate: Rotor Air Channels` in the **Label** text field.

3 Locate the **Rotation** section. In the **Angle** text field, type `range((360/APnr-360/APnr*APfct)/2,360/APnr,360)`.

4 Select the object **fil1** only.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

6 Click **Build Selected**.

Difference 1 (dif1)

In the **Model Builder** window, right-click **Difference 1 (dif1)** and choose **Duplicate**.

Difference: Rotor Core

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Difference 2 (dif2)**.

2 In the **Settings** window for **Difference**, type `Difference: Rotor Core` in the **Label** text field.

3 Locate the **Difference** section. Click to select the **Activate Selection** toggle button for **Objects to add**.

4 Select the object **spl1(12)** only.

5 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

6 Select the objects **rot1(1)**, **rot1(2)**, **rot1(3)**, **rot1(4)**, and **rot1(5)** only.

7 Select the **Keep objects to subtract** check box.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

9 Click **Build Selected**.

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** check box.

5 Click **Build Selected**.

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 `inf`.

5 In the **Inner radius** text field, type `30.5/2*0.99`.

Circle 1 (c1), Circle 2 (c2), Difference 1 (dif1), Difference: Rotor Core (dif2), Fillet 1 (fil1), Internal Rotor – Surface Mounted Magnets 1 (pi1), Rotate: Rotor Air Channels (rot1), Rotating Boundaries (disksel1), Split 1 (spl1), 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)**, **Split 1 (spl1)**, **Circle 1 (c1)**, **Circle 2 (c2)**, **Difference 1 (dif1)**, **Fillet 1 (fil1)**, **Rotate: Rotor Air Channels (rot1)**, **Difference: Rotor Core (dif2)**, **Union: Rotor (uni1)**, and **Rotating Boundaries (disksel1)**.

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.
Arkkio_toggle	1	1	Toggle Arkkio air gap - (1/0) (on/off)

4 Locate the **Domain Selections** section. In the table, select the **Keep** check boxes for **Stator iron**, **Stator slots**, and **All domains**.

5 Click **Build Selected**.

Scale 1 (sca1)

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 **Home** toolbar, click f(x) **Functions** and choose **Local>Ramp**.

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

3 In the **Location** text field, type `t_ramp/2`.

4 Click to expand the **Smoothing** section.

5 Select the **Size of transition zone at start** check box. In the associated text field, type `t_ramp`.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
alpha	w_rot*2*pi*rm1(t)[s]		Rotation angle

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 box, in the **Selections to add** list, choose **Rotor Magnets (Internal Rotor – Surface Mounted Magnets 1)** and **Difference: Rotor Core**.

5 Click **OK**.

6 Right-click **Structural Domains** and choose **Duplicate**.

Deforming Domains

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Selections** click **Structural Domains 1**.

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, select **Difference: Rotor Core**.

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 box, select **Rotor air (Internal Rotor – Surface Mounted Magnets 1)** in the **Selections to add** list.

8 Click **OK**.

9 In the **Settings** window for **Union**, locate the **Input Entities** section.

10 Under **Selections to add**, click **Add**.

11 In the **Add** dialog box, select **Rotate: Rotor Air Channels** in the **Selections to add** list.

12 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 box, 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 FROM LIBRARY

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 **Nonlinear Magnetic>Silicon Steel NGO>Silicon Steel NGO 35PN440**.

5 Click **Add to Component** in the window toolbar.

6 In the tree, select **AC/DC>Magnetic Materials (Bomatec)>NdFeB>BMN-42**.

7 Click **Add to Component** in the window toolbar.

8 In the tree, select **Built-in>High-strength alloy steel**.

9 Click **Add to Component** in the window toolbar.

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

MATERIALS

Silicon Steel NGO 35PN440 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silicon Steel NGO 35PN440 (mat2)**.

2 Select Domains 2 and 35 only.

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

4 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	1	Young's modulus and Poisson's ratio
Density	rho	7700	kg/m³	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 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`	1	Young's modulus and Poisson's ratio
Density	rho	`7600`	kg/m³	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 **Rotation speed** list, choose **User defined**. In the α text field, type `alpha`.

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 326 only.

South 1

1 In the **Model Builder** window, click **South 1**.

2 Select Boundary 323 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 `Ipk`.

5 In the α_i text field, type `init_ang+2*pi*f_el*rm1(t)[s]`.

6 From the **Winding layout configuration** list, choose **Automatic three phase**.

7 In the n_{poles} text field, type `Np`.

8 In the n_{slots} text field, type `Ns`.

9 In the **Number of coils per slot** text field, type `2`.

10 Locate the **Homogenized Multiturn Conductor** section. From the **Coil wire cross-section area** list, choose **Filling factor**.

11 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 1–29 and 31–46 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. In the α text field, type `alpha`.

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 30 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `1.3`.

8 Click **Build All**.

STUDY 1

Step 2: Time Dependent

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent>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 Find the **Algebraic variable settings** subsection. In the **Fraction of initial step for Backward Euler** text field, type `0.01`.

5 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**.

6 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.

7 From the **Nonlinear method** list, choose **Automatic (Newton)**.

8 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** check box. 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** check box.

5 Clear the **Solution** check box.

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 **1D Plot Group**.

2 In the **Settings** window for **1D 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** check box.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
`X-component`

8 Right-click **Point Graph 1** and choose **Duplicate**.

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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:

Legends
`Y-component`

Displacement

1 In the **Model Builder** window, click **Displacement**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **Two y-axes** check box.

4 In the table, select the **Plot on secondary y-axis** check box for **Point Graph 2**.

5 Select the **x-axis label** check box.

6 Select the **y-axis label** check box.

7 Select the **Secondary y-axis label** check box.

Follow the instructions below to plot the axial torque as shown in Figure 5.

Torque

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

2 In the **Settings** window for **1D 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** check box.

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 **1D Plot Group**, locate the **Axis** section.

3 Select the **Manual axis limits** check box.

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 Norm (rmm), Torque

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Magnetic Flux Density Norm (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`.

4 Right-click **Displacement (mbd)** and choose **Duplicate**.

Magnetic Flux Density Norm (rmm)

1 In the **Model Builder** window, under **Results>Export** click **Displacement (mbd) 1**.

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 Norm (rmm)**.