Electromagnetic and Mechanical Analysis of an Interior Permanent Magnet Motor

Introduction

The magnets in an Interior Permanent Magnet (IPM) motor are embedded in the rotor core, where they form narrow regions known as bridges. The thickness of the magnetic bridge is an important parameter to consider in design, both from the electromagnetic and the mechanical perspectives. As the magnetic saturation in bridge areas affects the electromagnetic characteristics of an IPM, the thickness of the bridge should be kept minimum to reduce losses. However, during high-speed rotation, these narrow bridges also experience high stress, caused by centrifugal forces. These two conflicting aspects makes the design and analysis of an IPM motor challenging.

As designers are often interested in the distribution of stresses and deformation in the stator and rotor, caused by the air gap forces, the interaction between electromagnetic and structural domains is an important aspect to consider. In this example, the coupling between the Solid Mechanics and Rotating Machinery, Magnetic interfaces for performing and mechanical and electromagnetic analysis of an IPM motor is demonstrated. The results give insight into magnetic flux density and stress distribution in the system.

Note: This model requires the AC/DC Module and the Structural Mechanics Module.

Model Definition

An IPM motor with 10 rotor poles and 12 stator slots is modeled in 2D. The diameter of the rotor and the stator are 60 mm and 100 mm respectively. The axial length of the motor is 40 mm. As shown in Figure 1, the magnets are embedded in a V-shaped configuration with an angle of 30°. The thickness of the permanent magnet is 2.5 mm. To reduce the weight and minimize material usage, rotor has some air cavities.

In this example, the size of the critical bridges is kept as 0.3 mm.

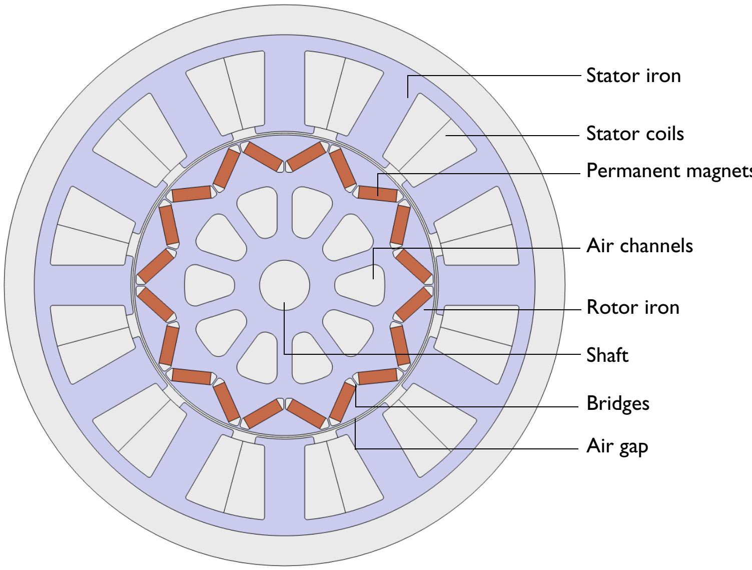

Figure 1: Geometry of the motor.

MAGNETIC-STRUCTURE INTERACTION

The interaction between electromagnetic and structural domains is modeled using a Magnetic–Elastic Interaction in Rotating Machinery interface. This interface consists of a Rotating Machinery, Magnetic interface, a Solid Mechanics 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 *Structural Mechanics Module User's Guide*.

ROTATING MACHINERY, MAGNETIC

The 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 soft iron with zero conductivity. The permanent magnets are made of Sintered NdFeB, creating a strong magnetic field. The rotational speed is taken as 20,000 rpm. The stator coil is excited with a peak current of 5 A, with an initial current angle for peak torque set as 200°.

SOLID MECHANICS

The Solid Mechanics interface is used for modeling the stator iron, rotor, and the embedded permanent magnets. For mechanical analysis, these parts are considered as elastic with properties as shown in Table 1.

TABLE 1: MECHANICAL PROPERTIES OF STATOR, ROTOR IRON AND MAGNETS.

Property	Symbol	Unit	Soft iron	Sintered NdFeB
Young's modulus	E	GPa	185	160
Poisson's ratio	ν	1	0.3	0.24
Mass density	ρ	kg/m^3	7500	7500

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

During high speed rotations, there are chances that the embedded permanent magnets in the rotor may get dislocated due to centrifugal force. As the connection between the magnets and the rotor influences the stress distribution, it is an important aspect to consider in modeling. Using a **Thin Layer** feature, the connection between the permanent magnets and rotor iron is modeled as a spring condition. These springs have high compressive and low tensile stiffness, effectively approximating a contact condition. The stiffness values of these springs are chosen in such a way that they are two orders of magnitude softer than the outside rotor material. The stiffness of the springs in tangential direction is large enough to prevent sliding of the magnets in the slots. The thickness of the layer is taken as 0.025 mm.

MOVING MESH

A deforming domain condition is assigned to the rotor air gap and other rotor air channels, which experience significant deformation due to the rotation of adjacent structural domains. For these domains, the shape 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** boundary condition is used to enable the sliding of the mesh.

STUDY

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

Results and Discussion

Figure 2 shows the von Mises stress in stator iron, rotor, and permanent magnets. Because of the cavities, high stresses are concentrated in the rotor near the bottom side of the air channels. Also, the inner bridges (those close to the rotor center) experience high stress levels. This is because the inner bridges support more rotor mass than the outer bridges.

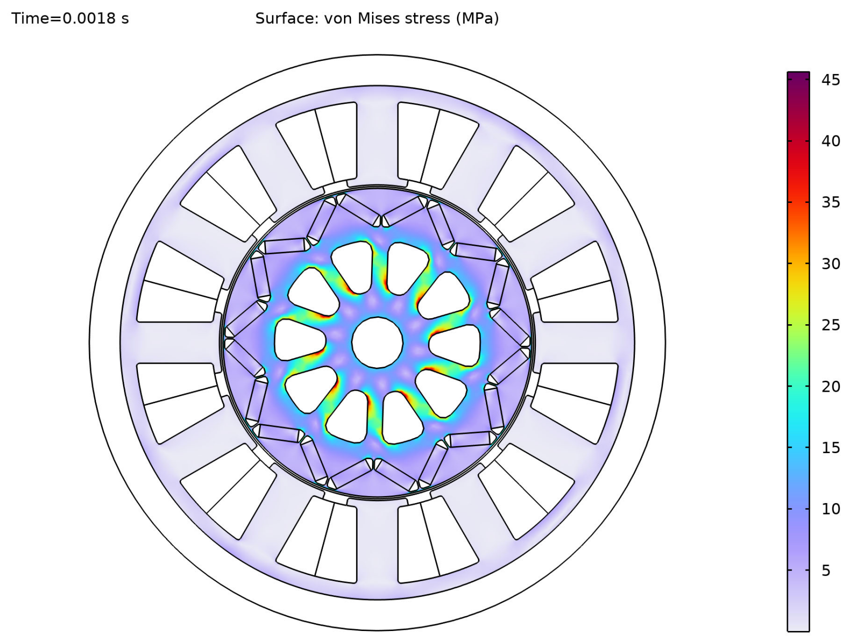

Figure 2: von Mises stress distribution in rotor and stator at $t = 0.0018$ *s.*

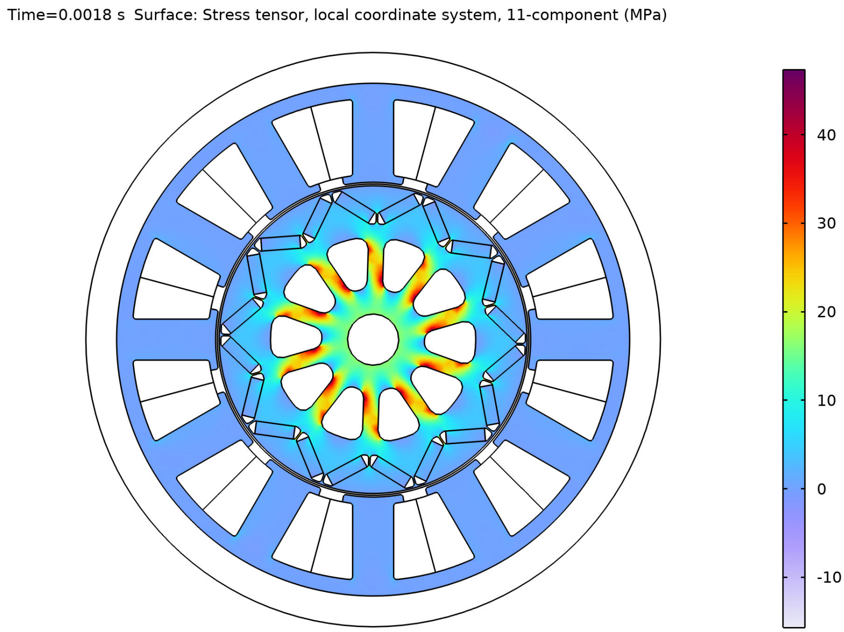

Figure 3: Stress distribution in radial direction at t = 0.0018 s.

Figure 3 shows the stress distribution in the radial direction in order to give insights about the tensile and compressive regions in the system. As seen from the figure, tensile stresses are mainly concentrated near the rotor air cavities and the bridges. As the magnets are prevented from lateral sliding, these areas are under compression.

The total displacement of stator iron, rotor, and permanent magnets is plotted in Figure 4. In Figure 5, the variation of the displacement of a sample point on the rotor as a function of time is plotted.

In Figure 6, the norm of the magnetic flux density and field lines are shown. Figure 7 plots the rotor torque ripple as a function of time for three electrical periods.

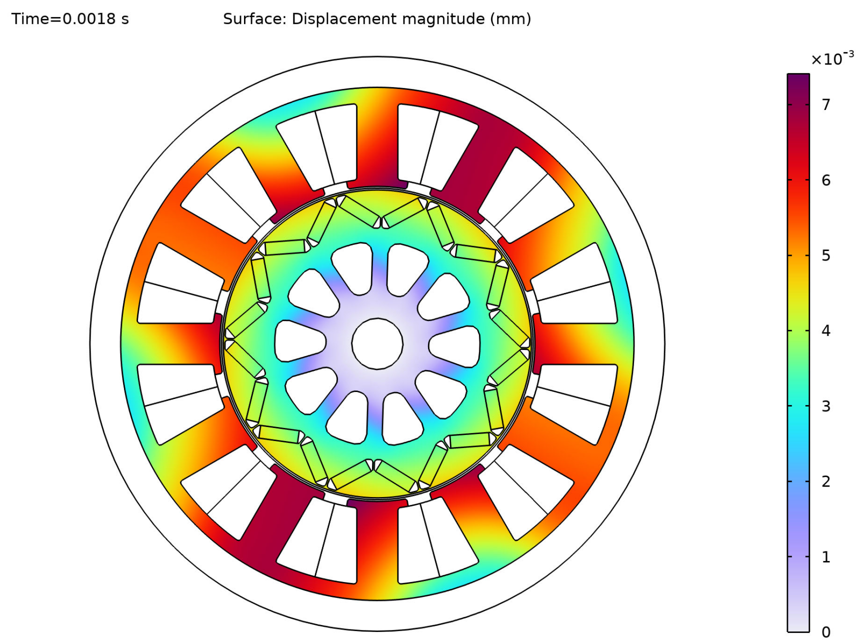

Figure 4: Displacement of the system at $t = 0.0018$ *s.*

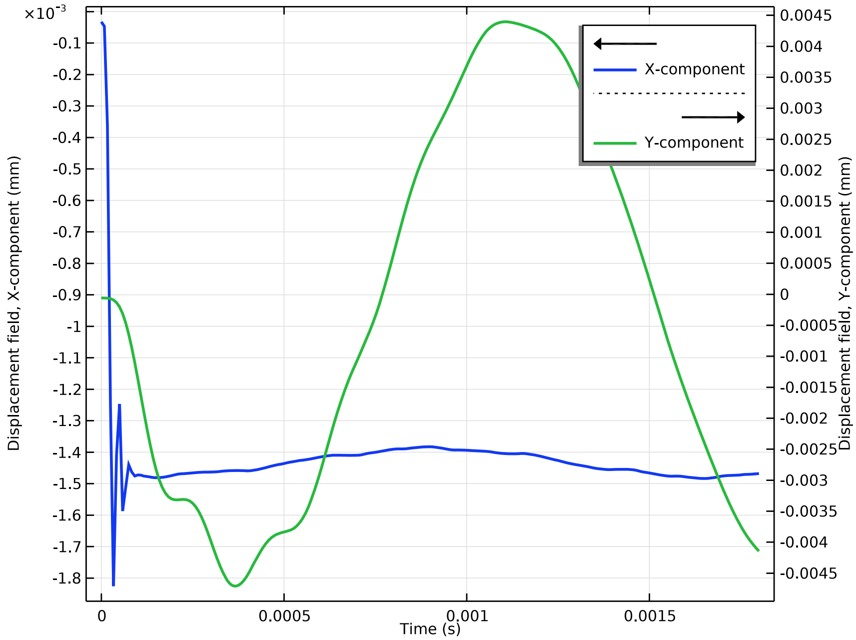

Figure 5: Displacement of a sample point on rotor as a function of time.

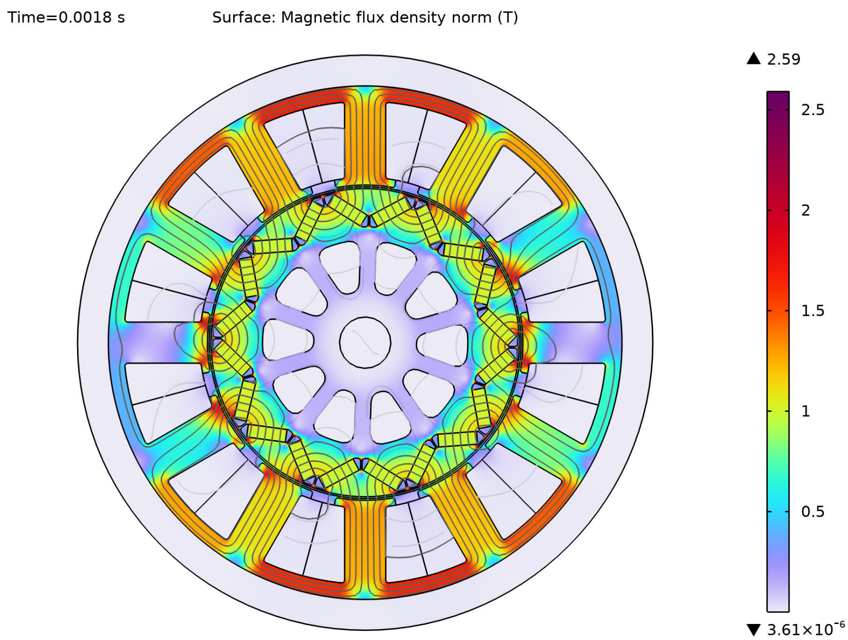

Figure 6: The norm and field lines of magnetic flux density at $t = 0.0018$ *s.*

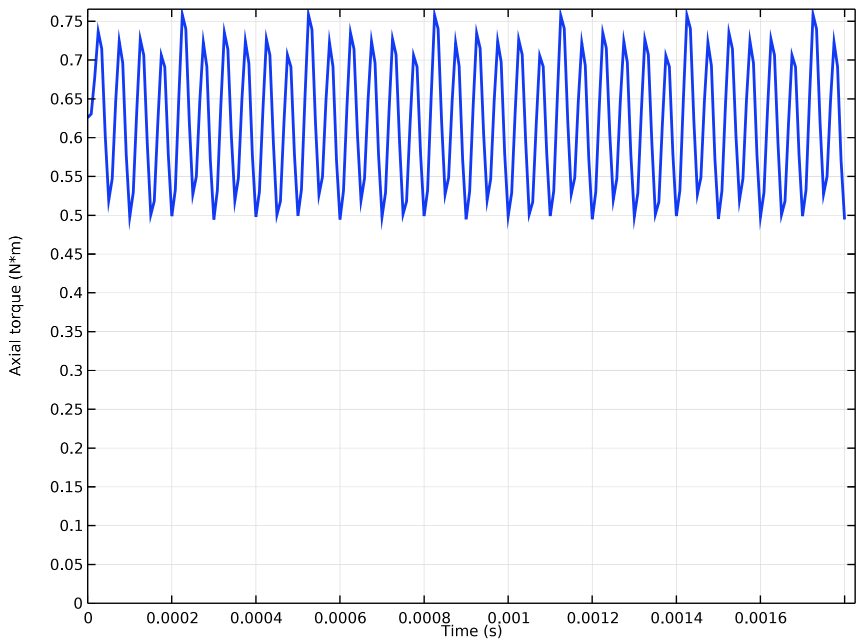

Figure 7: Rotor torque plotted as a function of time for three electrical periods.

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/interior_pm_motor_stress_analysis`

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–Elastic 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 `interior_pm_motor_stress_analysis_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>embedded_magnet_v_shape_internal_rotor_2d** in the tree.
4 Click **Add to Geometry**.

GEOMETRY 1

Internal Rotor – V-shaped Embedded Magnets 1 (pi1)

1. In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Internal Rotor – V-shaped Embedded 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
shaft_diam	`d_s`	10 mm	Diameter of the shaft
rotor_diam	`d_r`	60 mm	Diameter of the rotor
cont_diam	`d_cont`	60.7 mm	Diameter of the stator-rotor continuity interface
magnet_h	`mag_h`	2.5 mm	Height of the magnets
rotor_bridge_size	`0.3[mm]`	0.3 mm	Size of the bridge between magnets and outer surface of rotor
poles_bridge_size	`0.3[mm]`	0.3 mm	Size of the bridge between poles
v_distance	`0.3[mm]`	0.3 mm	Distance between magnets at the V-corner
flux_barrier_indent_fraction	`0`	0	Indent fraction of the flux barrier (set to 1 to not draw flux barrier)

4. Click to expand the **Domain Selections** section. In the table, select the **Keep** check boxes for **Shaft**, **Rotor air gap**, **Rotor_magnets**, **Rotor iron**, **Flux barriers**, **Rotor air**, **Rotor solid domains**, and **All domains**.
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 20.
4 Click **Build Selected**.
5 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 10.
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**.

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 30.
4 In the **Height** text field, type 3.
5 Locate the **Position** section. From the **Base** list, choose **Center**.
6 In the **x** text field, type 15.
7 Click **Build Selected**.

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.
2 In the **Settings** window for **Rotate**, locate the **Rotation** section.
3 In the **Angle** text field, type `range((360/Np)/2,360/Np,360-360/Np/2)`.
4 Select the object **r1** only.

5 Click **Build Selected**.

Difference 2 (dif2)

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

2 Select the object **dif1** 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 objects **rot1(1)**, **rot1(10)**, **rot1(2)**, **rot1(3)**, **rot1(4)**, **rot1(5)**, **rot1(6)**, **rot1(7)**, **rot1(8)**, and **rot1(9)** only.

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

7 Click **Build Selected**.

Difference 3 (dif3)

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

2 Select the object **spl1(2)** 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 From the **Objects to subtract** list, choose **Difference 2**.

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

7 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**.

Rotor Air Pocket Points

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

2 In the **Settings** window for **Adjacent Selection**, type `Rotor Air Pocket Points` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog box, select **Flux barriers (Internal Rotor – V-shaped Embedded Magnets 1)** in the **Input selections** list.

5 Click **OK**.

6 In the **Settings** window for **Adjacent Selection**, locate the **Output Entities** section.

7 From the **Geometric entity level** list, choose **Adjacent points**.

8 Click **Build Selected**.

Rotor Air Channel Points

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

2 In the **Settings** window for **Adjacent Selection**, type `Rotor Air Channel Points` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog box, select **Difference 2** in the **Input selections** list.

5 Click **OK**.

6 In the **Settings** window for **Adjacent Selection**, locate the **Output Entities** section.

7 From the **Geometric entity level** list, choose **Adjacent points**.

8 Click **Build Selected**.

9 In the **Model Builder** window, right-click **Rotor Air Channel Points (adjsel2)** and choose **Duplicate**.

Rotor Magnet Points

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Rotor Air Channel Points 1 (adjsel3)**.

2 In the **Settings** window for **Adjacent Selection**, type `Rotor Magnet Points` in the **Label** text field.

3 Click **Build Selected**.

4 Locate the **Input Entities** section. In the **Input selections** list, select **Difference 2**.

5 Click **Delete**.

6 Click **Add**.

7 In the **Add** dialog box, select **Rotor_magnets (Internal Rotor – V-shaped Embedded Magnets 1)** in the **Input selections** list.

8 Click **OK**.

Rotor Air Pocket Fillet Points

1 In the **Geometry** toolbar, click **Selections** and choose **Difference Selection**.

2 In the **Settings** window for **Difference Selection**, type `Rotor Air Pocket Fillet Points` in the **Label** text field.

3 Click **Build Selected**.

4 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Point**.

5 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

6 In the **Add** dialog box, select **Rotor Air Pocket Points** in the **Selections to add** list.

7 Click **OK**.

8 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

9 Click the **Add** button for **Selections to subtract**.

10 In the **Add** dialog box, select **Rotor Magnet Points** in the **Selections to subtract** list.

11 Click **OK**.

Fillet 1 (fil1)

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

2 In the **Settings** window for **Fillet**, locate the **Points** section.

3 From the **Vertices to fillet** list, choose **Rotor Air Pocket Fillet Points**.

4 Locate the **Radius** section. In the **Radius** text field, type `0.5[mm]`.

5 Click **Build Selected**.

Fillet 2 (fil2)

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

2 In the **Settings** window for **Fillet**, locate the **Points** section.

3 From the **Vertices to fillet** list, choose **Rotor Air Channel Points**.

4 On the object **fil1**, select Points 88–91, 116, 117, 132, 133, 135, 136, 145, 146, 167, 168, 172, 173, 182–185, 210–213, 222, 223, 227, 228, 249, 250, 259, 260, 262, 263, 278, 279, and 304–307 only.

5 Locate the **Radius** section. In the **Radius** text field, type `2[mm]`.

6 Click **Build Selected**.

Circle 1 (c1), Circle 2 (c2), Difference 1 (dif1), Difference 2 (dif2), Difference 3 (dif3), Fillet 1 (fil1), Fillet 2 (fil2), Internal Rotor – V-shaped Embedded Magnets 1 (pi1), Rectangle 1 (r1), Rotate 1 (rot1), Rotor Air Channel Points (adjsel2), Rotor Air Pocket Fillet Points (difsel1), Rotor Air Pocket Points (adjsel1), Rotor Magnet Points (adjsel3), Split 1 (spl1), Union: Rotor (uni1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1**, Ctrl-click to select **Internal Rotor** – **V-shaped Embedded Magnets 1 (pi1)**, **Split 1 (spl1)**, **Circle 1 (c1)**,

Circle 2 (c2), **Difference 1 (dif1)**, **Rectangle 1 (r1)**, **Rotate 1 (rot1)**, **Difference 2 (dif2)**, **Difference 3 (dif3)**, **Union: Rotor (uni1)**, **Rotor Air Pocket Points (adjsel1)**, **Rotor Air Channel Points (adjsel2)**, **Rotor Magnet Points (adjsel3)**, **Rotor Air Pocket Fillet Points (difsel1)**, **Fillet 1 (fil1)**, and **Fillet 2 (fil2)**.

2 Right-click and choose **Group**.

Rotor

In the **Settings** window for **Group**, type `Rotor` 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
backiron_th	`3[mm]`	3 mm	Thickness of back-iron
stator_diam	`d_st`	100 mm	Diameter of the stator
external_air_size	`6[mm]`	6 mm	Size of air external to stator
cont_diam	`d_cont`	60.7 mm	Diameter of the stator-rotor continuity interface
shoe_w	`12[mm]`	12 mm	Width of the shoe
tooth_h	`17[mm]-airgap`	16.3 mm	Height of the tooth

Name	Expression	Value	Description
tooth_w	8[mm]	8 mm	Width of the tooth
slot_outer_fillet_size	0.5[mm]	0.5 mm	Radius of the outer slot fillet
slot_inner_fillet_size	0.5[mm]	0.5 mm	Radius of the inner slot fillet
slot_winding_type	2	2	Slot winding type: 1-No partition, 2-Radial partition, 3-Azimuthal partition, 4-Radial and azimuthal partition.
Arkkio_toggle	2	2	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**.

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`.

Next, add selections to assign materials and physics features.

Rotor Structural Domains

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Rotor 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 – V-shaped Embedded Magnets 1)** and **Rotor iron (Internal Rotor – V-shaped Embedded Magnets 1)**.

5 Click **OK**.

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

Structural Domains

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

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, select **Stator iron (External Stator – Slotted 1)** in the **Selections to add** list.

5 Click **OK**.

Deforming Domains

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Deforming Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click + **Add**.

4 In the **Add** dialog box, select **Difference 2** in the **Selections to add** list.

5 Click **OK**.

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

7 Under **Selections to add**, click + **Add**.

8 In the **Add** dialog box, select **Flux barriers (Internal Rotor – V-shaped Embedded Magnets 1)** in the **Selections to add** list.

9 Click **OK**.

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

11 Under **Selections to add**, click + **Add**.

12 In the **Add** dialog box, select **Rotor air gap (Internal Rotor – V-shaped Embedded Magnets 1)** in the **Selections to add** list.

13 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 – V-shaped Embedded Magnets 1)** in the **Input selections** list.

5 Click **OK**.

6 Right-click **Shaft Boundaries** and choose **Duplicate**.

Rotor Magnet Boundaries

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

2 In the **Settings** window for **Adjacent**, type `Rotor Magnet Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. In the **Input selections** list, select **Shaft (Internal Rotor – V-shaped Embedded Magnets 1)**.

4 Under **Input selections**, click **Delete**.

5 Under **Input selections**, click **Add**.

6 In the **Add** dialog box, select **Rotor_magnets (Internal Rotor – V-shaped Embedded Magnets 1)** in the **Input selections** list.

7 Click **OK**.

8 Right-click **Rotor Magnet Boundaries** and choose **Duplicate**.

Rotor Air Pocket Boundaries

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Selections** click **Rotor Magnet Boundaries 1**.

2 In the **Settings** window for **Adjacent**, type `Rotor Air Pocket Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. In the **Input selections** list, select **Rotor_magnets (Internal Rotor – V-shaped Embedded Magnets 1)**.

4 Under **Input selections**, click **Delete**.

5 Under **Input selections**, click **Add**.

6 In the **Add** dialog box, select **Flux barriers (Internal Rotor – V-shaped Embedded Magnets 1)** in the **Input selections** list.

7 Click **OK**.

Magnet-Rotor Iron Contact Boundaries

1 In the **Definitions** toolbar, click **Difference**.

2 In the **Settings** window for **Difference**, type `Magnet-Rotor Iron Contact Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, select **Rotor Magnet Boundaries** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference**, locate the **Input Entities** section.

8 Under **Selections to subtract**, click **Add**.

9 In the **Add** dialog box, select **Rotor Air Pocket Boundaries** in the **Selections to subtract** list.

10 Click **OK**.

Rotor Magnet Boundaries

In the **Model Builder** window, right-click **Rotor Magnet Boundaries** and choose **Duplicate**.

Rotor Air Gap Boundaries

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Selections** click **Rotor Magnet Boundaries 1**.

2 In the **Settings** window for **Adjacent**, type `Rotor Air Gap Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. In the **Input selections** list, select **Rotor_magnets (Internal Rotor – V-shaped Embedded Magnets 1)**.

4 Under **Input selections**, click **Delete**.

5 Under **Input selections**, click **Add**.

6 In the **Add** dialog box, select **Rotor air gap (Internal Rotor – V-shaped Embedded Magnets 1)** in the **Input selections** list.

7 Click **OK**.

Rotating Boundaries

1 In the **Model Builder** window, right-click **Selections** and choose **Disk**.

2 In the **Settings** window for **Disk**, type `Rotating Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

5 Under **Selections**, click **Add**.

6 In the **Add** dialog box, select **Rotor Air Gap Boundaries** in the **Selections** list.

7 Click **OK**.

8 In the **Settings** window for **Disk**, locate the **Size and Shape** section.

9 In the **Outer radius** text field, type `d_cont/2+airgap/4`.

10 In the **Inner radius** text field, type `d_cont/2-airgap/4`.

11 In the **Model Builder** window, right-click **Rotating Boundaries** and choose **Duplicate**.

Boundary Layer Boundaries

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

2 In the **Settings** window for **Disk**, type `Boundary Layer Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Entities** list, choose **All**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `d_cont/2+airgap/3*4`.

5 In the **Inner radius** text field, type `d_cont/2-airgap/4*3`.

6 In the **Inner radius** text field, type `d_cont/2-airgap/4*3`.

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 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 **AC/DC>Soft Iron (Without Losses)**.

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

6 In the tree, select **AC/DC>Hard Magnetic Materials> Sintered NdFeB Grades (Chinese Standard)>N42 (Sintered NdFeB)**.

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

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

MATERIALS

Soft Iron (Without Losses) (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Soft Iron (Without Losses) (mat2)**.

2 Select Domains 2 and 29 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	185[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	7500	kg/m³	Basic

N42 (Sintered NdFeB) (mat3)

1 In the **Model Builder** window, click **N42 (Sintered NdFeB) (mat3)**.

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

3 From the **Selection** list, choose **Rotor_magnets (Internal Rotor – V-shaped Embedded 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	7500	kg/m³	Basic

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, 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`.

DEFINITIONS

Cylindrical System 2 (sys2)

1 In the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.

2 In the **Settings** window for **Cylindrical System**, locate the **Coordinate Names** section.

3 From the **Frame** list, choose **Material (X, Y, Z)**.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

Rotating Frame 1

1 In the **Model Builder** window, click **Rotating Frame 1**.

2 In the **Settings** window for **Rotating Frame**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Rotor Structural Domains**.

4 Locate the **Rotating Frame** section. From the **Rotation speed** list, choose **User defined**. In the α text field, type `alpha`.

Rigid Motion Suppression 1

1 In the **Physics** toolbar, click **Domains** and choose **Rigid Motion Suppression**.

2 In the **Settings** window for **Rigid Motion Suppression**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Stator iron (External Stator – Slotted 1)**.

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**.

Thin Layer 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 In the **Settings** window for **Thin Layer**, locate the **Boundary Properties** section.

3 In the L_{th} text field, type `tl`.

4 Locate the **Thin Layer** section. From the **Approximation** list, choose **Spring**.

5 Locate the **Boundary Selection** section. From the **Selection** list, choose **Magnet-Rotor Iron Contact Boundaries**.

Spring Material 1

1 In the **Physics** toolbar, click **Attributes** and choose **Spring Material**.

2 In the **Settings** window for **Spring Material**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Magnet-Rotor Iron Contact Boundaries**.

4 Locate the **Spring** section. From the **Spring type** list, choose **Total force as function of extension**.

5 Specify the $\mathbf{F}_{\text{tot}}(\mathbf{u}_{\text{e}})$ vector as

kt1*solid.tl1.uelt1	t1
kn*solid.tl1.ueln	n

DEFINITIONS

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
kt	5e11[N/m]	N/m	Spring stiffness in tension
kc	5e13[N/m]	N/m	Spring stiffness in compression
kn	kt*(solid.tl1.ueln>=0)+ kc*(solid.tl1.ueln<0)	N/m	Stiffness of thin elastic layer in normal direction
kt1	1e15[N/m]	N/m	Stiffness of thin elastic layer in tangential direction

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`.

4 Click to expand the **Discretization** section. From the **Magnetic vector potential** list, choose **Linear**.

Ampère's Law 2

1 In the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domains 2 and 29 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 – V-shaped Embedded 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 In the **Settings** window for **North**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 394 and 414 only.

South 1

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

2 Select Boundaries 390 and 422 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 Click the **Zoom Box** button in the **Graphics** toolbar.

3 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

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

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

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 **Curvature factor** text field, type `0.9`.

Size 1

1 In the **Model Builder** window, right-click **Mesh 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 From the **Selection** list, choose **Stator iron (External Stator – Slotted 1)**.

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 `1.5`.

8 Right-click **Size 1** and choose **Move Up**.

Free Triangular 1

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

Boundary Layers 1

1 In the **Mesh** toolbar, click  **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.

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

4 Select Domains 10 and 99 only.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Boundary Layer Boundaries**.

4 Locate the **Layers** section. In the **Number of layers** text field, type `1`.

5 In the **Thickness adjustment factor** text field, type 5.

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 0.1.

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 von Mises stress as shown in Figure 2.

von Mises Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `von Mises Stress (solid)` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **von Mises Stress (solid)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

Follow the instructions below to plot the radial stress as shown in Figure 3.

von Mises Stress (solid)

In the **Model Builder** window, right-click **von Mises Stress (solid)** and choose **Duplicate**.

Radial Stress (solid)

1 In the **Model Builder** window, under **Results** click **von Mises Stress (solid) 1**.

2 In the **Settings** window for **2D Plot Group**, type `Radial Stress (solid)` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Radial Stress (solid)** node, then click **Surface 1**.

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)>Solid Mechanics>Stress>Stress tensor, local coordinate system - N/m²>solid.slGp11 - Stress tensor, local coordinate system, 11-component**.

Follow the instructions below to plot system displacement as shown in Figure 4.

von Mises Stress (solid)

In the **Model Builder** window, under **Results** right-click **von Mises Stress (solid)** and choose **Duplicate**.

Displacement (solid)

1 In the **Model Builder** window, under **Results** click **von Mises Stress (solid) 1**.

2 In the **Settings** window for **2D Plot Group**, type `Displacement (solid)` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Displacement (solid)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `solid.disp`.

Follow the instructions below to plot the displacement of a sample point on rotor as shown in Figure 5.

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 264 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 7.

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**.

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, Displacement (solid), Radial Stress (solid), Stress, Thin Layer (solid), von Mises Stress (solid)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **von Mises Stress (solid)**, **Stress, Thin Layer (solid)**, **Radial Stress (solid)**, **Displacement (solid)**, and **Displacement**.

2 Right-click and choose **Group**.

Structural Plots

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

Radial Stress (solid)

In the **Model Builder** window, right-click **Radial Stress (solid)** and choose **Move Up**.

von Mises Stress (solid)

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, type `von Mises Stress (solid)` in the **Label** text field.

3 Locate the **Frames** section. In the **Number of frames** text field, type `50`.

4 Right-click **von Mises Stress (solid)** and choose **Duplicate**.

Magnetic Flux Density Norm (rmm)

1 In the **Model Builder** window, under **Results>Export** click **von Mises Stress (solid) 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)**.