Model created in COMSOL Multiphysics 6.4

Permanent Magnet Motor in 2D

Introduction

This tutorial model shows how to set up a three-phase permanent magnet motor simulation in 2D, using motor parts that are available in the AC/DC Module Part Library. The model consists of three studies. First, a stationary study solves the problem with a direct current, through the angular span of one pole pair, using Arkkio's method to calculate the rotor torque. Then, by specifying the initial mechanical angle to yield maximum torque, a transient study solves the time-dependent problem for a complete electrical period, and calculates the results for torque ripple and radial magnetic flux density. Finally, using the results from the transient study, the loss density in the stator iron is calculated with a frequency domain study.

Modeling

This model is set up in 2D and simulates the cross section on the rotational axis of the PM motor. The relevant equation is

$$\nabla \times \left(\frac{1}{\mu} \nabla \times \mathbf{A}\right) = \mathbf{J}$$

where $\mathbf{A}$ is the magnetic vector potential which defines the magnetic flux density $\mathbf{B} = \nabla \times \mathbf{A}$, $\mathbf{J}$ is the current density, and μ is the magnetic permeability. The equation is solved for the out-of-plane vector component only, which implies that in-plane currents and out-of-plane magnetic fields are neglected. This is a justified assumption for the 2D model which greatly simplifies and stabilizes the problem.

The separate rotor and stator objects are built as an assembly, and the relative rotation between rotor and stator is handled by the **Rotating Domain** node of the **Moving Mesh** feature, which includes all effects of relative motion between the parts. The domains are connected in the physics via boundary conditions on the continuity pair boundary, which resides in the air gap between them. This continuity pair allows for mesh discontinuities across the boundary where variables can be interpolated between the two independent meshes, ensuring continuity in the magnetic vector potential.

The torque is computed with the **Arkkio Torque Calculation** feature, which is automatically applied on the air gap domain adjacent to the continuity pair. The losses in the rotor and stator iron are calculated using a **Loss Calculation** subnode and a **Time to Frequency Losses** study. In the coils, the losses are Ohmic, while the losses in the iron are computed with the Steinmetz loss model.

Results and Discussion

The objective of the first study is to find the initial mechanical angle which produces the maximum torque on the rotor. As seen in Figure 1, the parametric sweep of the initial angle yields a curve displaying two extremes: one corresponding to accelerating torque, and the other corresponding to deceleration in the direction of the prescribed counter-clockwise rotation. For the maximum accelerating torque, the former is chosen for the initial angle. The subsequent transient study solves the synchronous rotation of the stator field and the rotor. Figure 2 plots the rotor torque ripple as a function of time for one electrical period. Finally, a **Time to Frequency Losses** study calculates the loss density using the results of the previous transient study. Figure 4 shows the resulting loss density in the stator as well as the rotor iron.

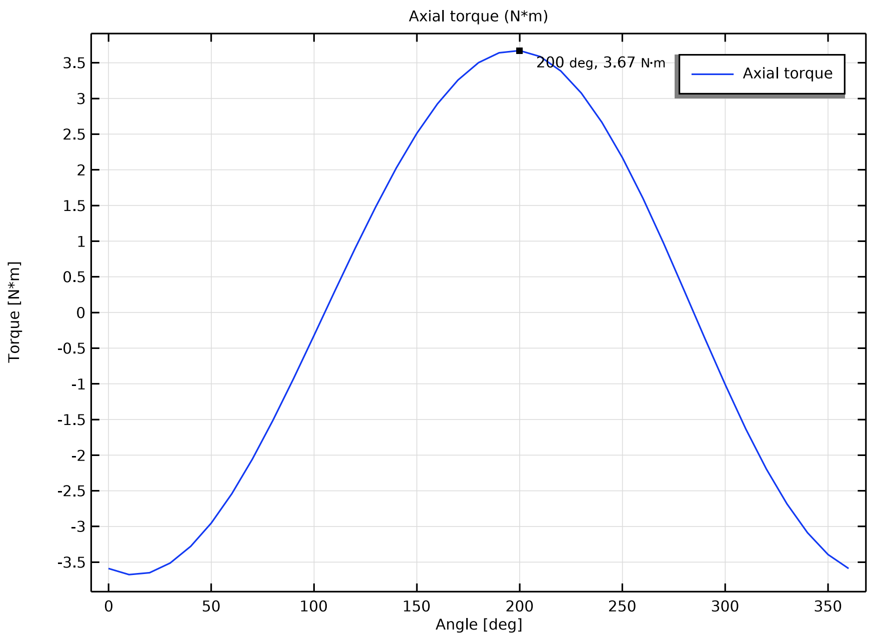


Figure 1: Rotor torque plotted as a function of the initial mechanical angle.

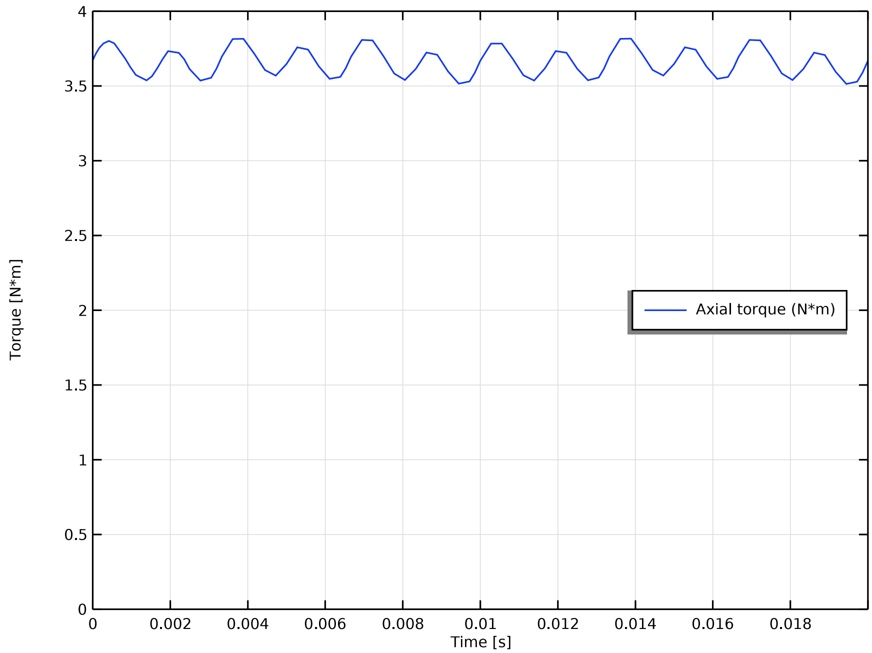


Figure 2: Rotor torque plotted as a function of time for a complete electrical period.

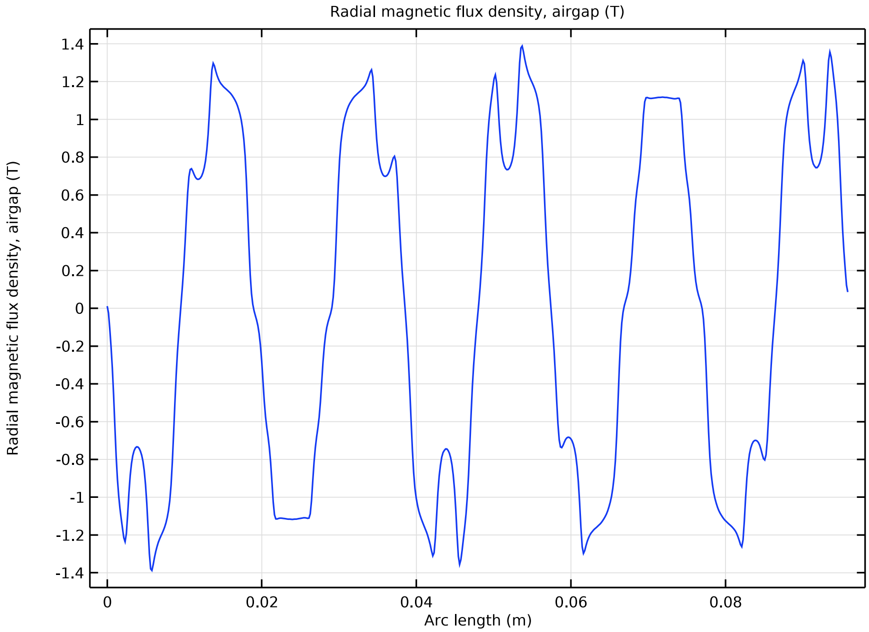


Figure 3: Radial magnetic flux density plotted versus the arc length of the continuity pair boundary, for time t = 0.

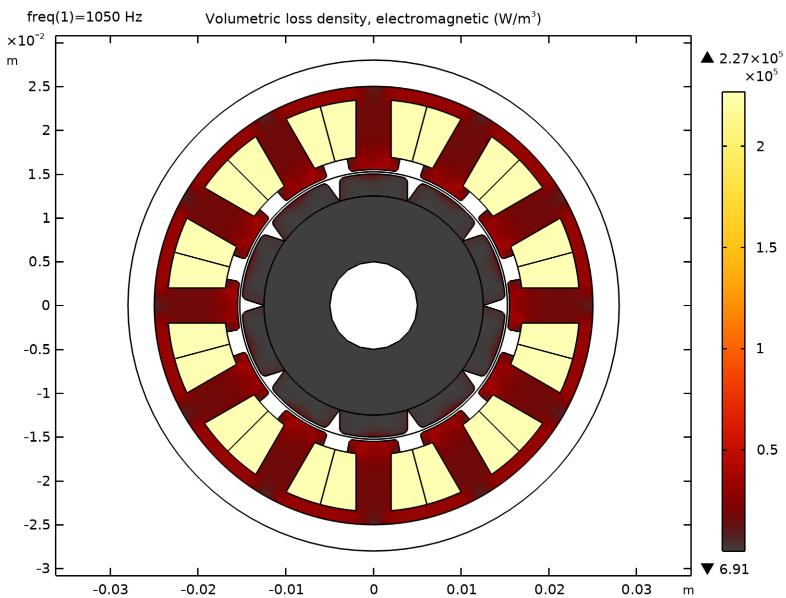


Figure 4: Loss density in the motor.

Application Library path: `ACDC_Module/Devices,_Motors_and_Generators/pm_motor_2d_introduction`

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 **AC/DC** > **Electromagnetics and Mechanics** > **Rotating Machinery, Magnetic (rmm)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Stationary**.

6 Click **Done**.

GEOMETRY 1

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 In the table, enter the following settings:

Name	Expression	Value	Description
L	200[mm]	0.2 m	Out-of-plane thickness of motor
init_ang	0[deg]	0 rad	Initial electrical angle
Np	10	10	Number of poles
Ns	12	12	Number of slots
w_rot	600[rpm]	10 1/s	Rotational speed
f_el	w_rot*(Np/2)	50 1/s	Electrical frequency
I0	10[A]	10 A	Peak current
Nturn	10	10	Number of wire turns in slot
ff_slot	0.8	0.8	Slot filling factor

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 **Part Libraries** window, select **AC/DC Module** > **Rotating Machinery 2D** > **Rotors** > **Internal** > **surface_mounted_magnet_internal_rotor_2d** in the tree.

3 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

4 Click to expand the **Domain Selections** section. In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Shaft	√	√	None
Rotor iron	√	√	None
Odd magnets		√	None
Even magnets		√	None
Rotor magnets	√	√	None
Rotor solid domains		√	None
Rotor air		√	None
All	√	√	None

5 Click to expand the **Boundary Selections** section. In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Exterior	√	√	None

PART LIBRARIES

1 In the **Home** toolbar, click **Windows** and choose **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.

4 Locate the **Domain Selections** section. In the table, enter the following settings:

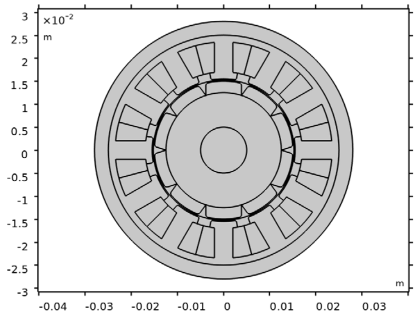
Name	Keep	Physics	Contribute to
Stator iron	√	√	None
Stator slots	√	√	None
Stator air		√	None
All	√	√	None

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 **Home** toolbar, click **Build All**.
5 Click the **Zoom Extents** button in the **Graphics** toolbar.

6 In the **Model Builder** window, click **Geometry 1**.

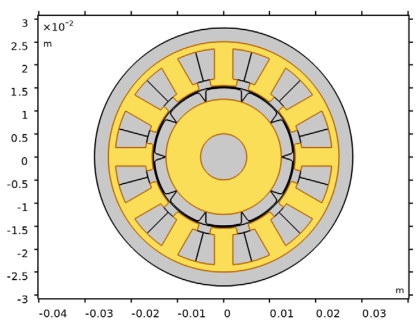


Create union selections for the motor iron parts.

DEFINITIONS

Iron

1. In the **Definitions** toolbar, click **Union**.
2. In the **Settings** window for **Union**, type `Iron` 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 iron (Internal Rotor – Surface Mounted Magnets 1)** and **Stator iron (External Stator – Slotted 1)**.
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 **AC/DC** > **Soft Iron (Without Losses)**.

6 Click the **Add to Component** button in the window toolbar.

7 In the tree, select **AC/DC** > **Copper**.

8 Click the **Add to Component** button in the window toolbar.

9 In the tree, select **AC/DC** > **Hard Magnetic Materials** > **Sintered NdFeB Grades (Chinese Standard)** > **N54 (Sintered NdFeB)**.

10 Click the **Add to Component** button in the window toolbar.

11 In the tree, select **Built-in** > **Iron**.

12 Click the **Add to Component** button in the window toolbar.

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

MATERIALS

Soft Iron (Without Losses) (mat2)

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

2 From the **Selection** list, choose **Iron**.

Iron (mat5)

1 In the **Model Builder** window, click **Iron (mat5)**.

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)**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	4000	1	Basic
Electric conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

Property	Variable	Value	Unit	Property group
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	`12.2e-6[1/K]`	1/K	Basic
Heat capacity at constant pressure	Cp	`440[J/(kg*K)]`	J/(kg·K)	Basic
Density	rho	`7870[kg/m^3]`	kg/m^3	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`76.2[W/(m*K)]`	W/(m·K)	Basic
Young's modulus	E	`200e9[Pa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.29`	1	Young's modulus and Poisson's ratio

Copper (mat3)

1 In the **Model Builder** window, click **Copper (mat3)**.

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

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

N54 (Sintered NdFeB) (mat4)

1 In the **Model Builder** window, click **N54 (Sintered NdFeB) (mat4)**.

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)**.

COMPONENT 1 (COMP1)

Rotating Domain 1

1 In the **Physics** toolbar, click **Moving Mesh** and choose **Rotating Domain**.

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

3 From the **Selection** list, choose **All domains (Internal Rotor – Surface Mounted Magnets 1)**.

4 Locate the **Rotation** section. From the **Rotation type** list, choose **Specified rotational velocity**.

5 In the ω text field, type `w_rot*2*pi`.

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`.

Laminated Core, Ampère's Law 1

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

2 In the **Settings** window for **Laminated Core, Ampère's Law**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Iron**.

4 Locate the **Lamination Model** section. From the **Lamination model** list, choose **Out-of-plane lamination**.

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**.

Loss Calculation 1

In the **Physics** toolbar, click **Attributes** and choose **Loss Calculation**.

North 1

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

2 Select Boundary 268 only.

South 1

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

2 Select Boundary 265 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 `I0`.

5 In the α_i text field, type `init_ang`.

6 In the f_t text field, type `f_el`.

7 Locate the **Homogenized Conductor** section. In the N text field, type `Nturn`.

8 In the f text field, type `ff_slot`.

9 Locate the **Multiphase Winding** section. From the **Winding layout configuration** list, choose **Automatic three phase**.

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

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

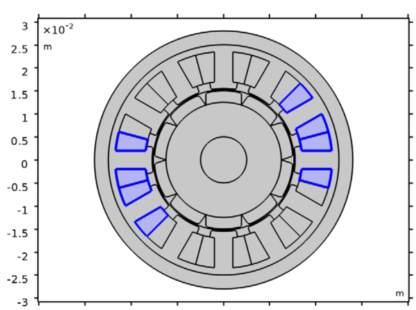
12 In the **Number of coils per slot** text field, type `2`.

13 Click **Add Phases**.

Automatic Phase 1

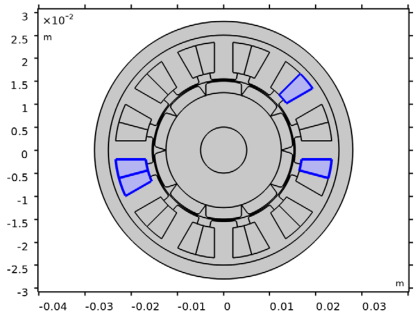
Selection of the generated phases can be inspected.

1 In the **Model Builder** window, click **Automatic Phase 1**.



Reversed Current Direction 1

In the **Model Builder** window, expand the **Automatic Phase 1** node, then click **Reversed Current Direction 1**.



Multiphase Winding 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Rotating Machinery, Magnetic (rmm)** click **Multiphase Winding 1**.

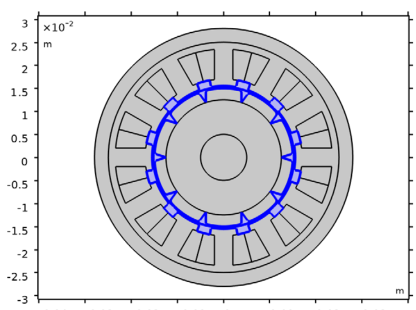
Loss Calculation 1

In the **Physics** toolbar, click **Attributes** and choose **Loss Calculation**.

Next, implement the Arkkio Torque Calculation feature for calculating the torque on the rotor. The node is automatically applied to the air gap. The Arkkio force integrand is multiplied with a support function which is nonzero in the correct radial extent, between the rotor magnets and stator iron.

Arkkio Torque Calculation 1

In the **Physics** toolbar, click **Domains** and choose **Arkkio Torque Calculation**.



Set up a probe for the motor torque.

DEFINITIONS

Torque

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type `Torque` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Rotating Machinery, Magnetic** > **Mechanical** > **rmm.Tark_1 - Axial torque - N·m**.

Adjust the default mesh to ensure sufficient resolution of magnetic field in the airgap and in stator core.

Identity Boundary Pair 1 (ap1)

1 In the **Model Builder** window, click **Identity Boundary Pair 1 (ap1)**.

2 In the **Settings** window for **Pair**, locate the **Source Boundaries** section.

3 Click **Create Selection**.

4 In the **Create Selection** dialog, type `source` in the **Selection name** text field.

5 Click **OK**.

6 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

7 Click **Create Selection**.

8 In the **Create Selection** dialog, type `dest` in the **Selection name** text field.

9 Click **OK**.

Airgap Boundaries

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

2 In the **Settings** window for **Union**, type `Airgap 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, in the **Selections to add** list, choose **source** and **dest**.

6 Click **OK**.

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, right-click **Mesh 1** and choose **Size**.

Size 2

Right-click **Mesh 1** and choose **Size**.

Size 1, Size 2

Right-click and choose **Move Up**.

Size 1

1 In the **Model Builder** window, click **Size 1**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **Airgap Boundaries**.
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 `0.5[mm]/3`.

Size 2

1 In the **Model Builder** window, click **Size 2**.
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** checkbox. In the associated text field, type `1[mm]`.
8 Click **Build All**.

Configure a stationary study to find the electrical angle providing maximum motoring torque.

STUDY 1: INITIAL ELECTRICAL ANGLE SWEEP

1 In the **Model Builder** window, click **Study 1**.
2 In the **Settings** window for **Study**, type `Study 1: Initial Electrical Angle Sweep` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1: Initial Electrical Angle Sweep** click **Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
init_ang (Initial electrical angle)	range(0,10[deg], 360[deg])	deg

6 In the **Model Builder** window, collapse the **Study 1: Initial Electrical Angle Sweep** node.

Monitor the torque while solving by clicking the **Probe Plot 1** next to the **Graphics** window after pushing the **Compute**-button.

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

RESULTS

Streamline 1

1 In the **Model Builder** window, expand the **Magnetic Flux Density (rmm)** node.

2 Right-click **Streamline 1** and choose **Disable**.

Contour 1

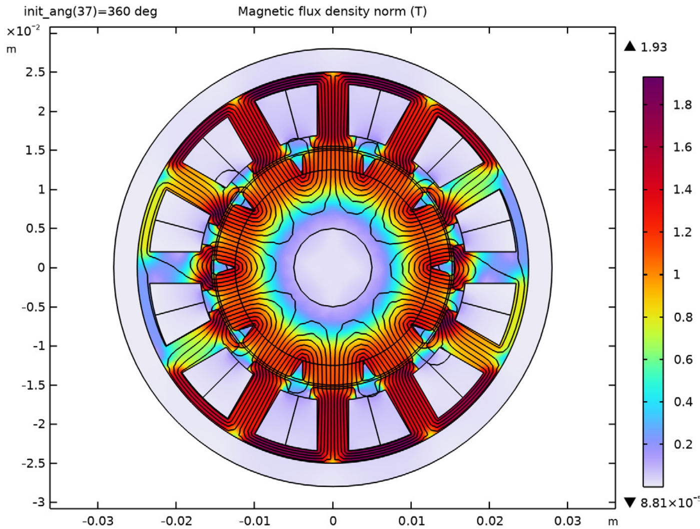
1 In the **Model Builder** window, click **Contour 1**.

2 In the **Settings** window for **Contour**, locate the **Levels** section.

3 In the **Total levels** text field, type 16.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

5 In the **Magnetic Flux Density (rmm)** toolbar, click **Plot**.



Torque

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 1**.

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

3 Locate the **Plot Settings** section.

4 Select the **x-axis label** checkbox. In the associated text field, type `Angle [deg]`.

5 Select the **y-axis label** checkbox. In the associated text field, type `Torque [N*m]`.

Torque Initial Electrical Angle Sweep

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

2 In the **Settings** window for **1D Plot Group**, type `Torque Initial Electrical Angle Sweep` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **x-axis label** checkbox. In the associated text field, type `Angle [deg]`.

5 Select the **y-axis label** checkbox. In the associated text field, type `Torque [N*m]`.

Global 1

1 Right-click **Torque Initial Electrical Angle Sweep** 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

Graph Marker 1

1 Right-click **Global 1** and choose **Graph Marker**.

2 In the **Settings** window for **Graph Marker**, locate the **Text Format** section.

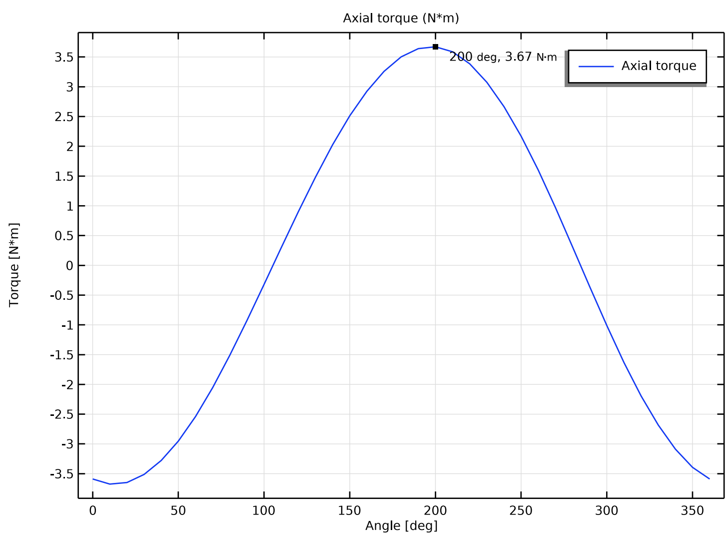
3 Select the **Show x-coordinate** checkbox.

4 Select the **Include unit** checkbox.

5 In the **Precision** text field, type 3.

6 Locate the **Display** section. From the **Display** list, choose **Max**.

7 In the **Torque Initial Electrical Angle Sweep** toolbar, click **Plot**.



The maximum torque is found at an initial electrical angle offset of 200°. Update init_ang with this value to orient the stator field with respect to rotor magnets so as to achieve maximum torque production.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
init_ang	200[deg]	3.4907 rad	Initial mechanical angle

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Stationary**.

4 Click the **Add Study** button in the window toolbar.

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

STUDY 2: SYNCHRONOUS ROTATION

In the **Settings** window for **Study**, type Study 2: Synchronous Rotation in the **Label** text field.

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(0,1/12/6,1.1)/f_el.

Solution 2 (sol2)

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

2 In the **Model Builder** window, collapse the **Study 2: Synchronous Rotation** node.

3 Click **Compute**.

RESULTS

Torque Ripple

1 In the **Model Builder** window, under **Results** click **Torque**.

2 In the **Settings** window for **1D Plot Group**, type Torque Ripple in the **Label** text field.

3 Locate the **Plot Settings** section. In the **x-axis label** text field, type Time [s].

4 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

5 In the **x minimum** text field, type 0.

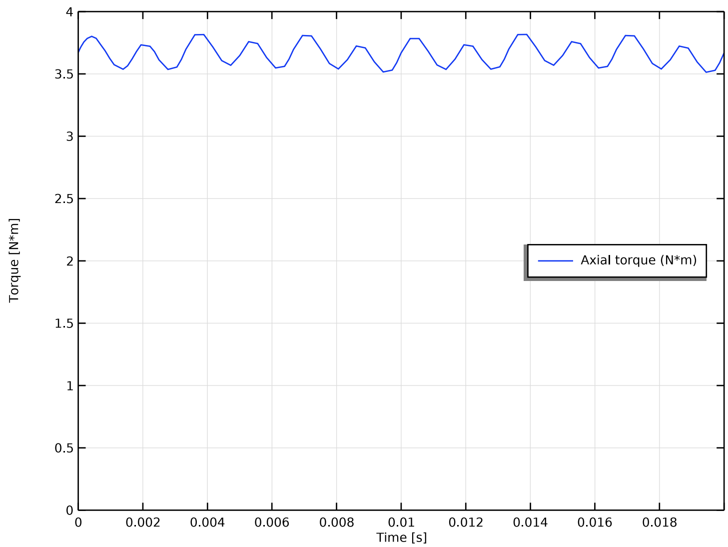
6 In the **x maximum** text field, type 0.02.

7 In the **y minimum** text field, type 0.

8 In the **y maximum** text field, type `4`.

9 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

10 In the **Torque Ripple** toolbar, click **Plot**.



Now plot the radial component of the magnetic flux density in the air gap. To do that, define a suitable boundary within the air gap, and plot the quantity along its arc length.

Air Gap Radial Magnetic Flux Density

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

2 In the **Settings** window for **1D Plot Group**, type `Air Gap Radial Magnetic Flux Density` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Synchronous Rotation/ Solution 2 (sol2)**.

4 From the **Time selection** list, choose **First**.

Line Graph 1

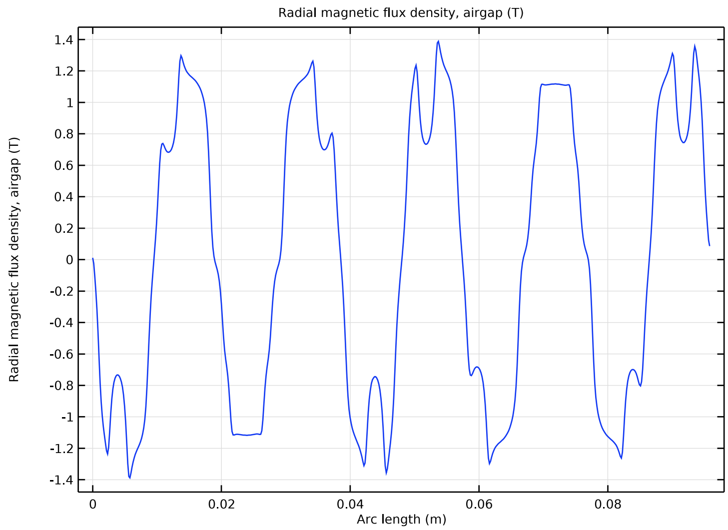
1 Right-click **Air Gap Radial Magnetic Flux Density** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 From the **Selection** list, choose **Exterior (Internal Rotor – Surface Mounted Magnets 1)**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `rmm.ark1.Brad`.

5 In the **Air Gap Radial Magnetic Flux Density** toolbar, click **Plot**.



ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces > Time-to-Frequency Losses**.

4 Click the **Add Study** button in the window toolbar.

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

STUDY 3: TIME AVERAGE LOSS CALCULATION

In the **Settings** window for **Study**, type `Study 3: Time Average Loss Calculation` in the **Label** text field.

Step 1: Time-to-Frequency Losses

1 In the **Model Builder** window, under **Study 3: Time Average Loss Calculation** click **Step 1: Time-to-Frequency Losses**.

2 In the **Settings** window for **Time-to-Frequency Losses**, locate the **Study Settings** section.

3 From the **Input study** list, choose **Study 2: Synchronous Rotation, Time Dependent**.

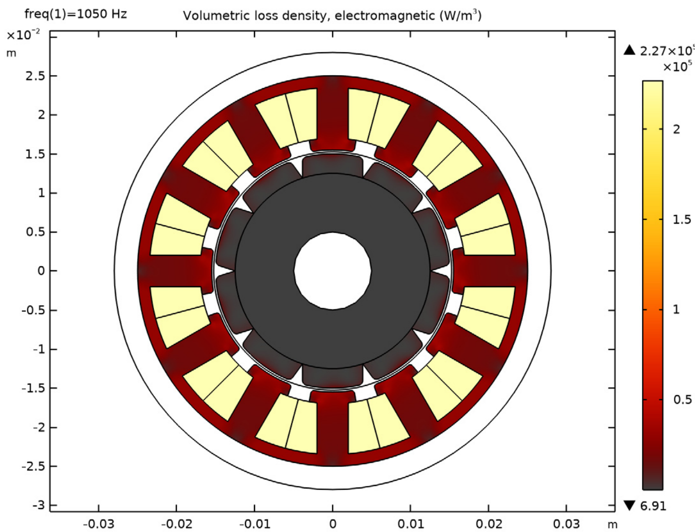
4 In the **Electrical period** text field, type `1/f_el`.

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

RESULTS

Cycle Averaged Losses (rmm)

In the **Cycle Averaged Losses (rmm)** toolbar, click **Plot**.



COMPONENT 1 (COMP1)

In the **Model Builder** window, collapse the **Component 1 (comp1)** node.

RESULTS

Torque over One Electrical Period

1. In the **Results** toolbar, click **Evaluation Group**.
2. In the **Settings** window for **Evaluation Group**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 2: Synchronous Rotation/Solution 2 (sol2)**.
4. In the **Label** text field, type `Torque over One Electrical Period`.

Global Evaluation 1

1. Right-click **Torque over One Electrical Period** and choose **Global Evaluation**.
2. In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
3. In the table, enter the following settings:

Expression	Unit	Description
`rmm.Tark_1`	`N*m`	`Axial torque`

Torque over One Electrical Period

1 In the **Model Builder** window, click **Torque over One Electrical Period**.

2 In the **Settings** window for **Evaluation Group**, locate the **Data** section.

3 From the **Time selection** list, choose **Interpolated**.

4 In the **Times (s)** text field, type `range(0.1,1/12/6,1.1)/f_el`.

5 In the **Torque over One Electrical Period** toolbar, click **Evaluate**.

Torque Harmonics

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

2 In the **Settings** window for **1D Plot Group**, type `Torque Harmonics` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Torque Harmonics`.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `Harmonic Order`.

Table Graph 1

1 Right-click **Torque Harmonics** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Source** list, choose **Evaluation group**.

4 From the **Transformation** list, choose **Discrete Fourier transform**.

5 From the **Show** list, choose **Frequency spectrum**.

6 From the **Scale** list, choose **Multiply by sampling period**.

7 Click to expand the **Preprocessing** section. Find the **x-axis column** subsection. From the **Transformation** list, choose **Linear**.

8 In the **Scaling** text field, type `f_el`.

9 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

10 In the **Torque Harmonics** toolbar, click **Plot**.