Created in COMSOL Multiphysics 6.1

# 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 $\mu$ 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.

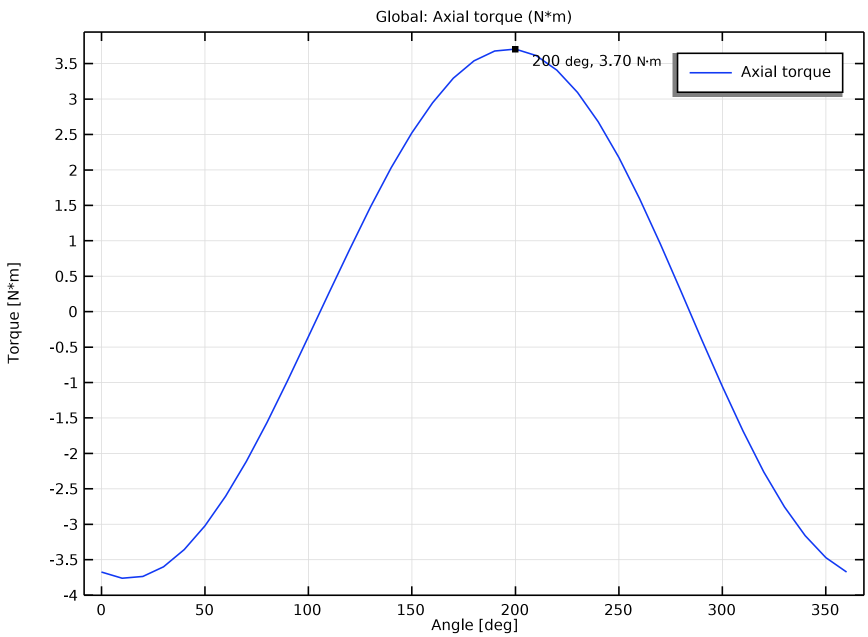



*Figure 1: Rotor torque plotted as a function of the initial mechanical angle.*

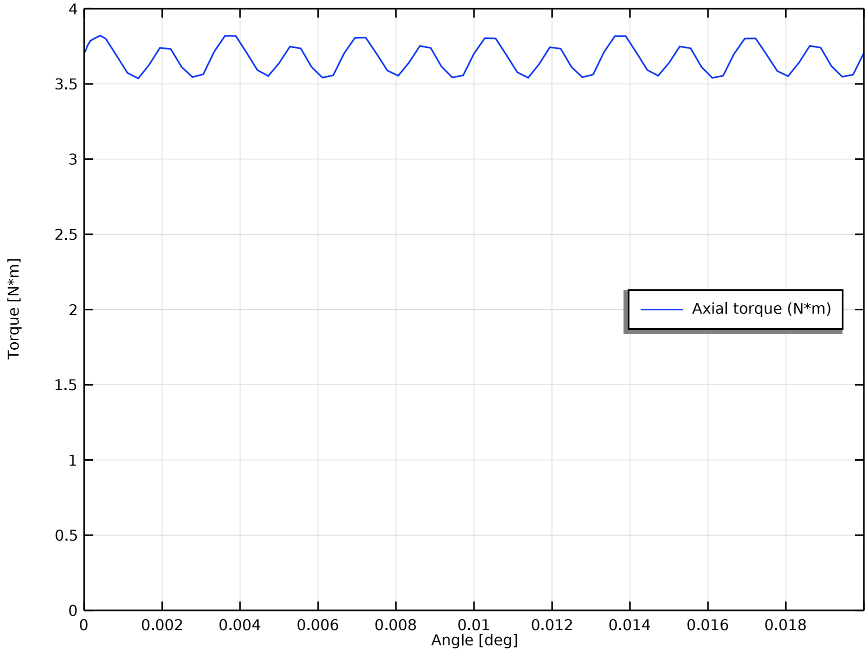



*Figure 2: Rotor torque plotted as a function of time for a complete electrical period.*

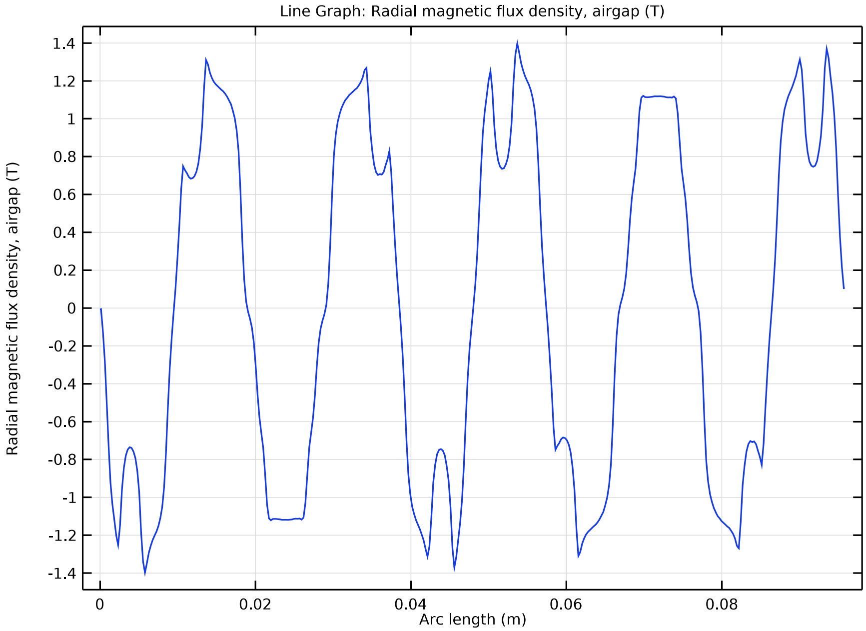



*Figure 3: Radial magnetic flux density plotted versus the arc length of the continuity pair boundary, for time t = 0.*

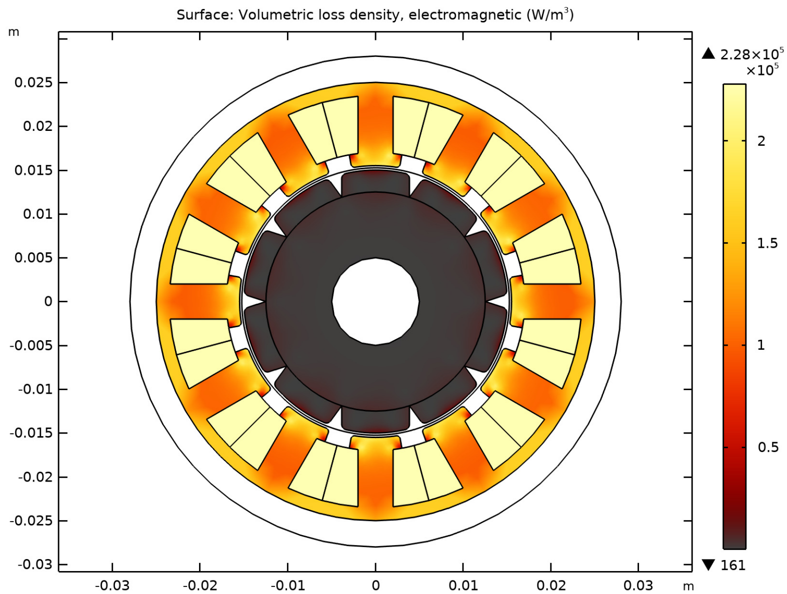



*Figure 4: Loss density in the motor.*

---

**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 pre-defined 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:

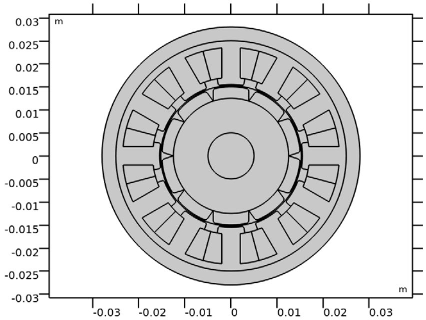
| Name | Keep | Physics | Contribute to |
|---|---|---|---|
| Stator iron | √ | √ | None |
| Stator slots | √ | √ | None |
| Stator air | | √ | None |
| All | √ | √ | None |

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.




Create union selections for the motor iron parts.

## DEFINITIONS

### *Iron*

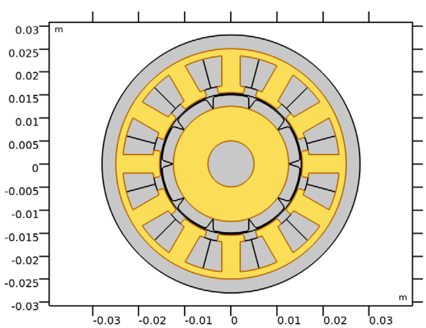
**1** In the **Definitions** toolbar, click **Union**.

**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 box, 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 **Home** 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 **Add to Component** in the window toolbar.

5 In the tree, select **AC/DC>Soft Iron (Without Losses)**.

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

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

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

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

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

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

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

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

3 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 |
| Electrical conductivity | sigma_iso ; sigmaii = sigma_iso, sigmaij = 0 | 0 | S/m | Basic |

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Relative permittivity | epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0 | 1 | 1 | Basic |
| 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 **Definitions** toolbar, click **Moving Mesh** and choose **Domains>Rotating Domain**.

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

3 From the **Selection** list, choose **All (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`.

#### *B-H Iron Regions*

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

2 In the **Settings** window for **Ampère's Law**, type `B-H Iron Regions` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Iron**.

4 Locate the **Constitutive Relation B-H** section. From the **Magnetization model** list, choose **B-H curve**.

#### *Loss Calculation 1*

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

2 In the **Settings** window for **Loss Calculation**, locate the **Loss Model** section.

3 From the **Loss model** list, choose **Steinmetz**.

#### *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**.

#### *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 Boundaries 266, 278, 280, and 286 only.

*South 1*

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

2 Select Boundaries 262, 264, 268, and 277 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 $\alpha_i$ text field, type `init_ang`.

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

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

8 From the **Coil wire cross-section area** list, choose **Filling factor**.

9 In the $f$ text field, type `ff_slot`.

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

11 In the **Number of poles** text field, type `Np`.

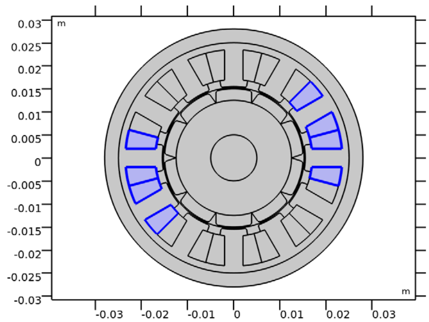
12 In the **Number of slots** text field, type `Ns`.

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

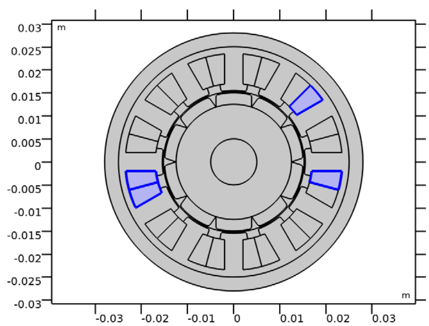
14 Click **Add Phases**.

*Automatic Phase 1*

Selection of the generated phases can be inspected.



*Reversed Current Direction 1*



*Multiphase Winding 1*

In the **Model Builder** window, expand the **Automatic Phase 1** node, then click **Component 1 (comp1)>Rotating Machinery, Magnetic (rmm)>Multiphase Winding 1**.

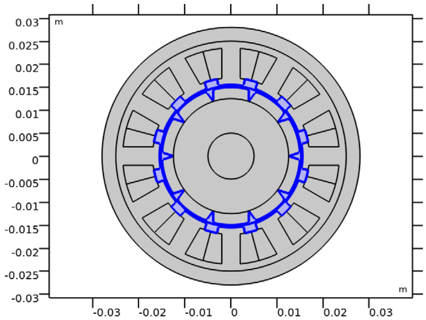
*Loss Calculation 1*

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 where torque will be calculated.

*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 box, 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 box, 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 box, in the **Selections to add** list, choose **source** and **dest**.

6 Click **OK**.

## MESH 1

*Size*

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

*Size 1*

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.

2 Drag and drop **Size 1** below **Size**.

3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Airgap boundaries**.

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

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

8 Select the **Maximum element size** check box. In the associated text field, type `0.5[mm]/3`.

9 Click **Build All**.

10 In the **Maximum element size** text field, type `0.5[mm]/2`.

11 Click **Build All**.

Set <l>Linear</l> elements for the discretization. This will yield a more reliable solution near regions of magnetic saturation.

## 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**, click to expand the **Discretization** section.

3 From the **Magnetic vector potential** list, choose **Linear**.

4 From the **Magnetic scalar potential** list, choose **Linear**.

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** check box.

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` |

### *Solution 1 (sol1)*

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1: Initial Electrical Angle Sweep>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

5 In the **Maximum number of iterations** text field, type `30`.

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

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

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

## RESULTS

### *Streamline 1*

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

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

### *Contour 1*

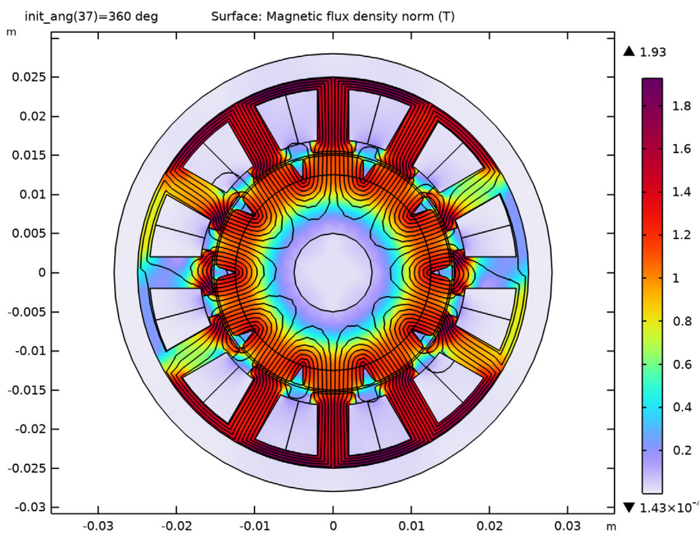
1 In the **Model Builder** window, click **Contour 1**.

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 Norm (rmm)** toolbar, click **Plot**.




### *1D Plot Group 3*

In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

### *Torque*

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 2**.

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** check box. In the associated text field, type `Angle [deg]`.

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

*Torque Initial Electrical Angle Sweep*

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

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** check box. In the associated text field, type `Angle [deg]`.

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

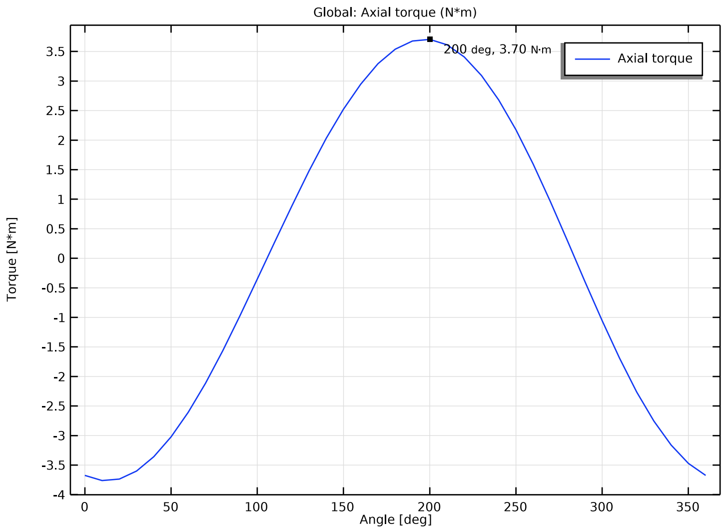
3 Select the **Show x-coordinate** check box.

4 Select the **Include unit** check box.

5 In the **Display 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 |

In order for the transient solver to achieve a stable solution of the nonlinear problem, set <l>Update Jacobian</l> to <l>On every iteration</l>. This will make the convergence more robust within each time step.

## 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 **Add Study** in the window toolbar.

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

## STUDY 2: SYNCHRONOUS ROTATION, TWO ELECTRICAL PERIODS

1 In the **Model Builder** window, click **Study 2**.

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

### *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(0,1/12/6,2)/f_el`.

### *Solution 2 (sol2)*

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2: Synchronous Rotation, Two Electrical Periods>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.

5 In the **Maximum number of iterations** text field, type `30`.

6 In the **Model Builder** window, expand the **Study 2: Synchronous Rotation, Two Electrical Periods>Solver Configurations>Solution 2 (sol2)>Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

7 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

8 From the **Jacobian update** list, choose **On every iteration**.

9 In the **Model Builder** window, collapse the **Study 2: Synchronous Rotation, Two Electrical Periods** node.

10 In the **Study** toolbar, 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 **Axis** section. Select the **Manual axis limits** check box.

**4** In the **x minimum** text field, type `0`.

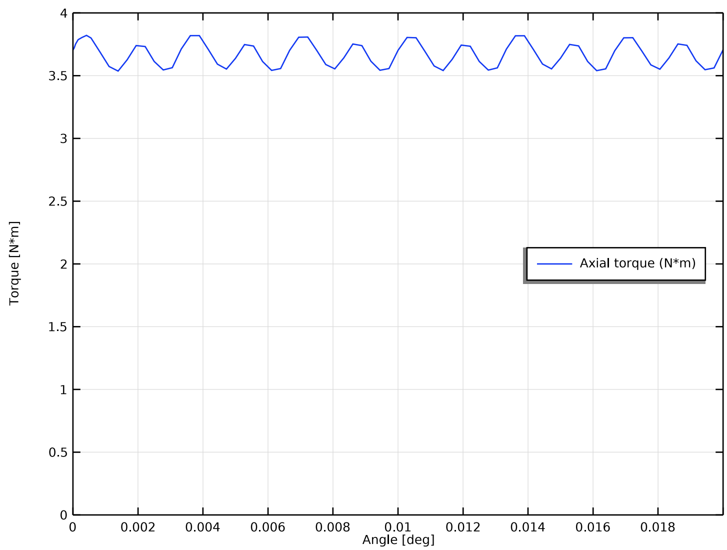
**5** In the **x maximum** text field, type `0.02`.

**6** In the **y minimum** text field, type `0`.

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

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

**9** 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 **Home** toolbar, click **Add Plot Group** and choose **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, Two Electrical Periods/Solution 2 (sol2)**.

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

### *Line Graph 1*

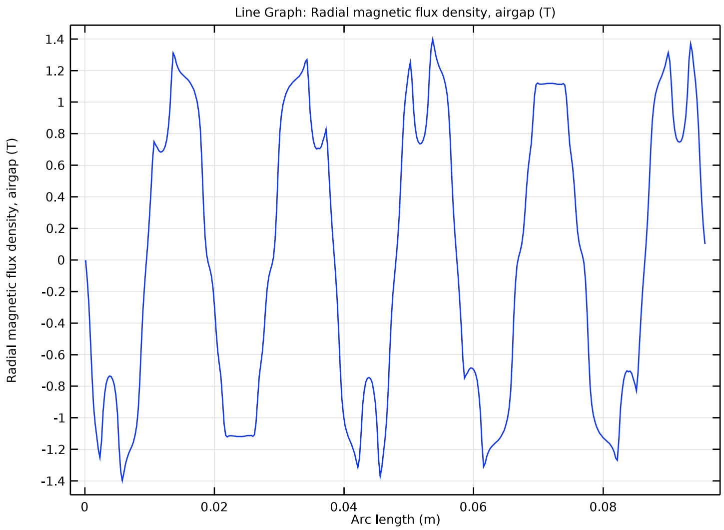
**1** Right-click **Air Gap Radial Magnetic Flux Density** and choose **Line Graph**.

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 **Add Study** in the window toolbar.

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

## STUDY 3: LOSS CALCULATION OVER ONE ELECTRICAL PERIOD

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study 3: Loss Calculation over One Electrical Period` in the **Label** text field.

### *Step 1: Time to Frequency Losses*

1 In the **Model Builder** window, under **Study 3: Loss Calculation over One Electrical Period** 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, Two Electrical Periods, Time Dependent**.

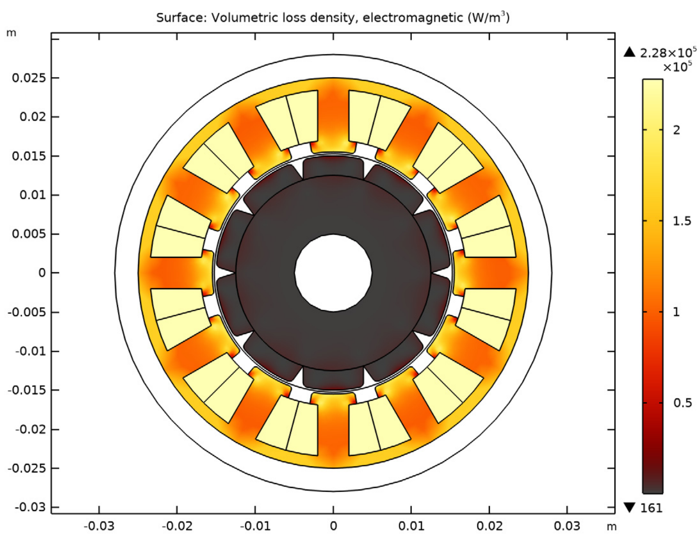
4 In the **Electrical period** text field, type `1/f_el`.

5 In the **Home** 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, Two Electrical Periods/ 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(1,1/12/6,2)/f_el`.

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

*Torque harmonics*

1 In the **Home** toolbar, click **Add Plot Group** and choose **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`.

*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 **Evaluation group** list, choose **Torque over One Electrical Period**.

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

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

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

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

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

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

11 In the **Torque harmonics** toolbar, click **Plot**.