Created in COMSOL Multiphysics 6.1

Permanent Magnet Motor in 3D

Introduction

Permanent magnet (PM) motors are used in many high end applications, for example in electric and hybrid vehicles. An important design limitation is that the permanent magnets are sensitive to high temperature. The eddy current losses in the steel/iron parts of the motor can easily be reduced by laminating these. However, due to manufacturing limitations, the permanent magnets cannot easily be laminated so the heating can be quite substantial as illustrated in this model.

Model Definition

An 18 pole permanent magnet motor is modeled in 3D. Sector symmetry and axial mirror symmetry is utilized to reduce the computational effort while capturing the full 3D behavior of the device. Figure 1 shows the full PM motor.

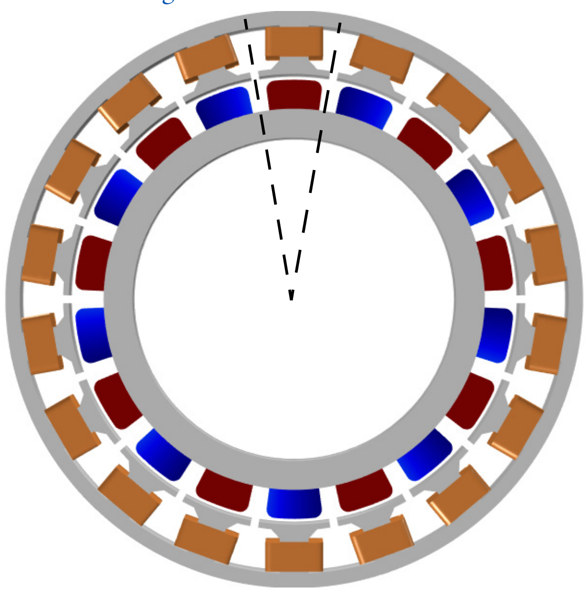


Figure 1: Drawing of the permanent magnet motor showing how the rotor and stator iron (gray), stator winding (Cu) and permanent magnets (blue/red depending on radial magnetization) are constructed. The antisymmetric sector is indicated by the dashed line. In addition mirror symmetry in the axial (out-of-plane) direction is utilized.

The conducting part of the rotor is modeled using Ampère's law:

$$\sigma\frac{\partial \mathbf{A}}{\partial t} + \nabla \times \left(\frac{1}{\mu}\nabla \times \mathbf{A}\right) = 0$$

whereas the nonconducting parts of both the rotor and stator are modeled using a magnetic flux conservation equation for the scalar magnetic potential:

$$-\nabla \cdot (\mu \nabla V_{\mathrm{m}} - \mathbf{B}_{\mathrm{r}}) = 0$$

Rotation is modeled using the ready-made physics interface for rotating machinery. The central part of the geometry, containing the rotor and part of the air-gap, is modeled as rotating relative to the coordinate system of the stator. The rotor and the stator are created as two separate geometry objects, so it is required to use an assembly (see the Geometry chapter in the *COMSOL Multiphysics Reference Manual* for details).

The electromagnetic losses in the magnets are computed with the Time to Frequency Losses study. This can later be used as a distributed, time-averaged, heat source in a separate heat transfer analysis (not included). The thermal time scale is typically much larger than the time variation of the eddy current losses so separating the electromechanical and thermal analyses is usually necessary for computational efficiency.

Results and Discussion

Figure 2 shows the magnetic flux density for the motor in it's stationary state, that is the initial conditions for the time-dependent simulation. In this state the coil current is zero.

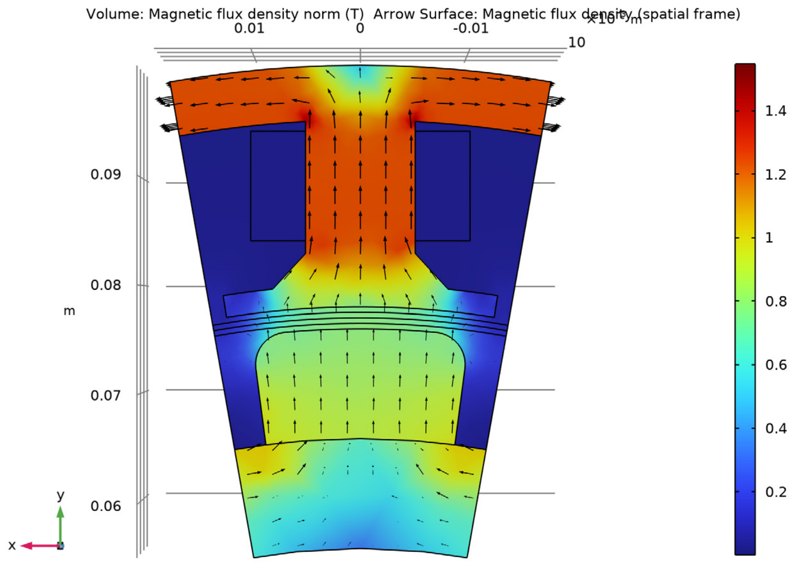


Figure 2: Magnetic flux density from the permanent magnets only with the rotor at rest.

Figure 3 shows the magnetic flux density for the motor after revolving one sector angle. In this plot the air and coil domains are excluded in order to get a better view.

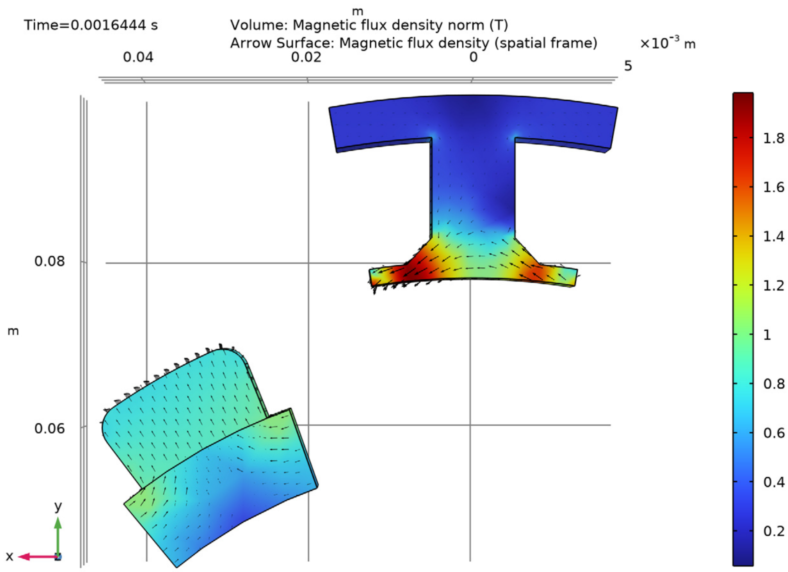


Figure 3: Magnetic flux density after revolving one sector angle.

Figure 4 shows the time evolution of the total eddy current losses in the magnet.

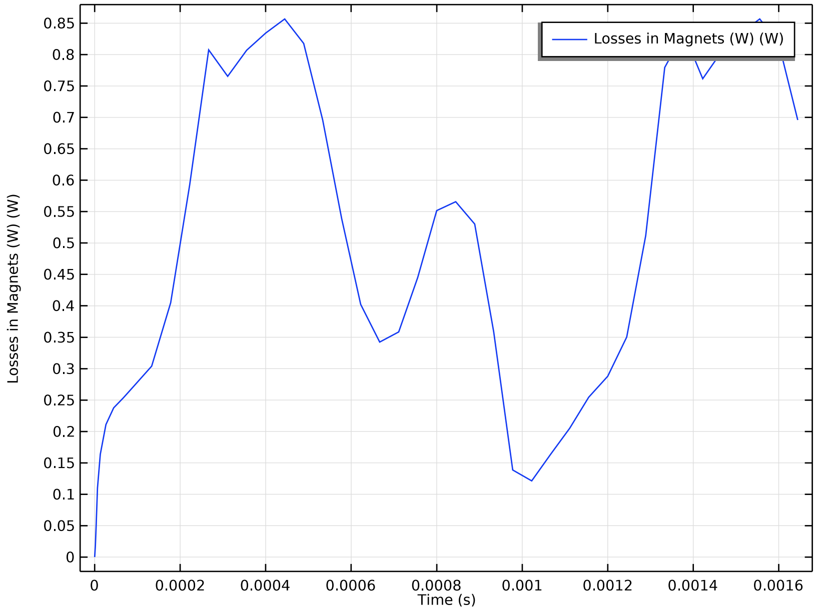


Figure 4: The eddy current loss in the magnets as a function of time.

[Figure 5](#) shows the time averaged eddy current loss power in the magnet in a period.

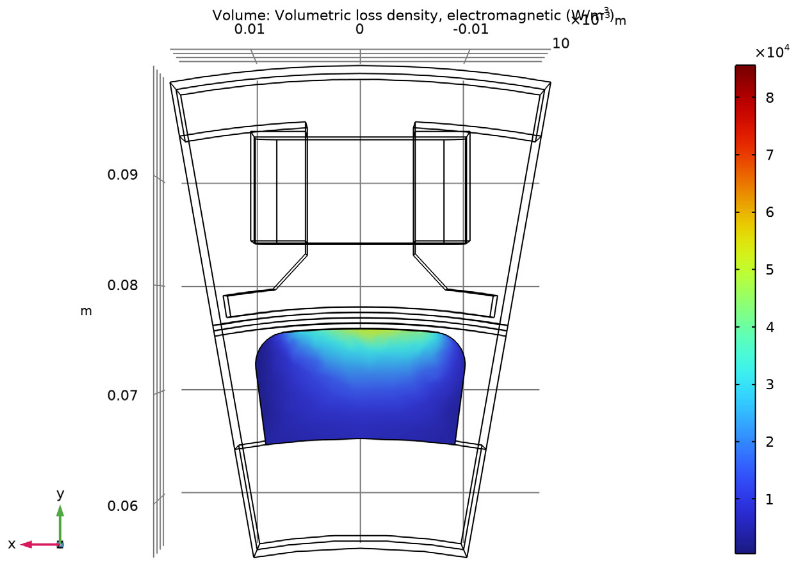


Figure 5: Time averaged eddy current loss power density in the magnet.

Application Library path: `ACDC_Module/Devices,_Motors_and_Generators/pm_motor_3d`

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 **3D**.

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**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `pm_motor_3d_parameters.txt`.

GEOMETRY 1

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `pm_motor_3d.mphbin`.

5 Click **Import**.

Form Union (fin)

An assembly must be used so that rotor and stator parts can be meshed independently.

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

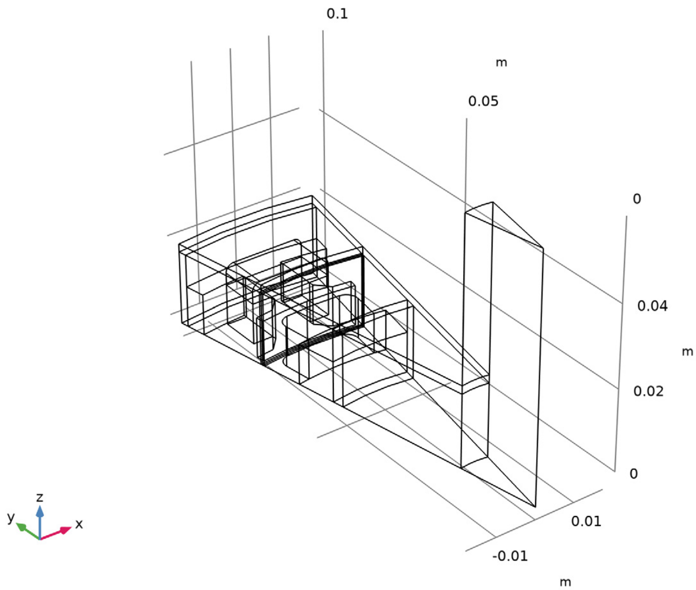
2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Select the **Create imprints** check box.

5 In the **Home** toolbar, click **Build All**.

6 Click the **Wireframe Rendering** button in the **Graphics** toolbar.



MATERIALS

Proceed to define materials.

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>Hard Magnetic Materials>Sintered NdFeB Grades (Chinese Standard)>N50 (Sintered NdFeB)**.

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

9 In the tree, select **Built-in>Aluminum**.

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

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

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

13 In the tree, select **Built-in>Copper**.

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

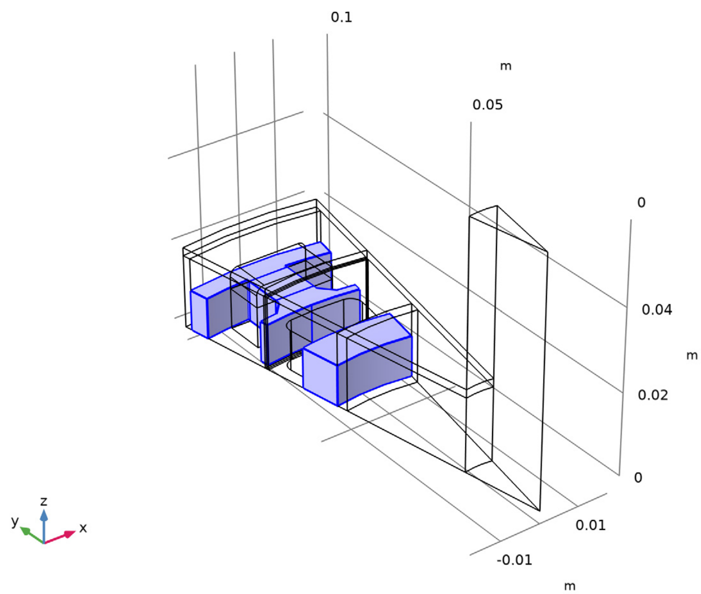
15 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

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 4 and 12 only.



For the easiest modeling setup a finite conductivity is added wherever the magnetic vector potential formulation is used.

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**
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`1[S/m]`	S/m	Basic

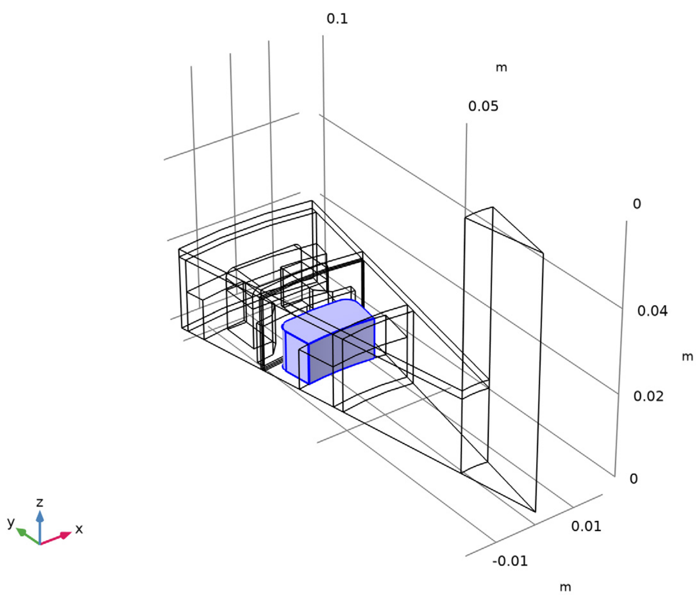
Custom the magnetic material of the permanent magnet.

Magnet

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **N50 (Sintered NdFeB) (mat3)**.

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

3 Select Domain 14 only.



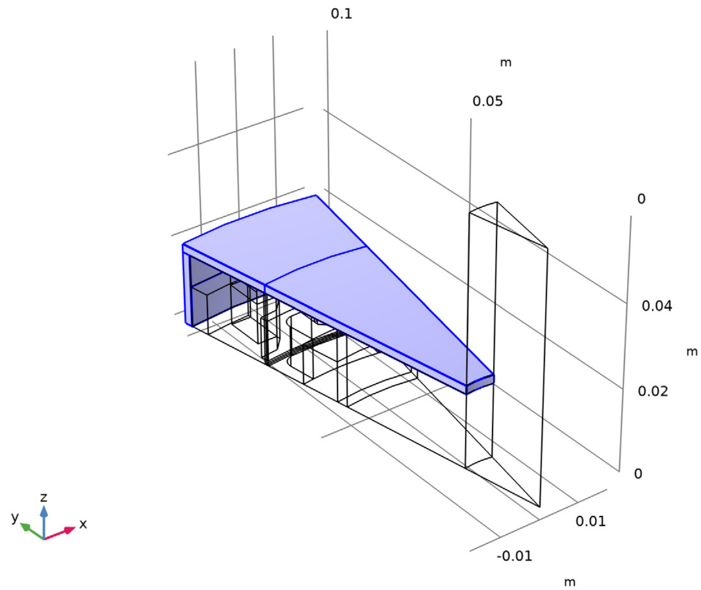
4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`7e5`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	`1`	1	Basic
Recoil permeability	murec_iso ; murecii = murec_iso, murecij = 0	`1.02`	1	Remanent flux density
Remanent flux density norm	normBr	`1[T]`	T	Remanent flux density

Aluminum (mat4)

1 In the **Model Builder** window, click **Aluminum (mat4)**.

2 Select Domains 1–3 only.

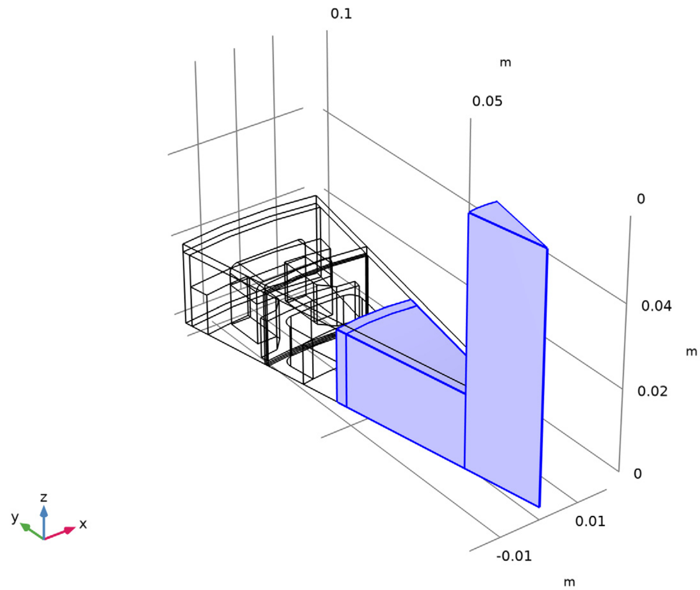


Some materials, like this one will not be used in the simulation but could be useful later, for example in a heat transfer simulation.

Structural steel (mat5)

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

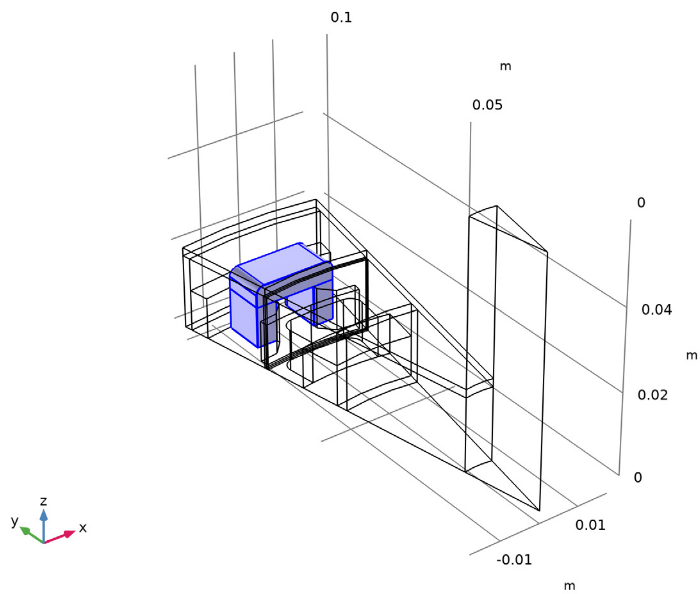
2 Select Domains 13, 15, and 16 only.



Copper (mat6)

1 In the **Model Builder** window, click **Copper (mat6)**.

2 Select Domains 7–9 only.

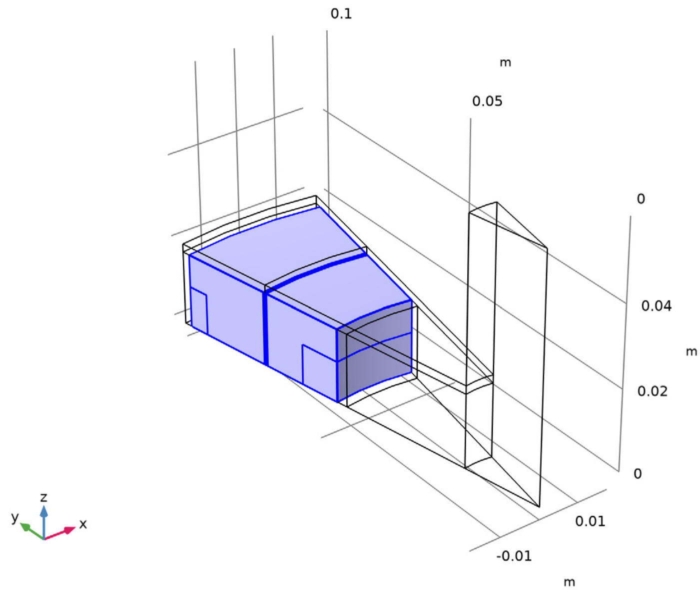


ROTATING MACHINERY, MAGNETIC (RMM)

Proceed to set up the physics. Limit the electromagnetic simulation to the relevant domains.

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

2 Select Domains 4–12 and 14 only.



Use linear elements in order to make the solution as fast and robust as possible.

3 In the **Settings** window for **Rotating Machinery, Magnetic**, click to expand the **Discretization** section.

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

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

Magnetic Flux Conservation - air

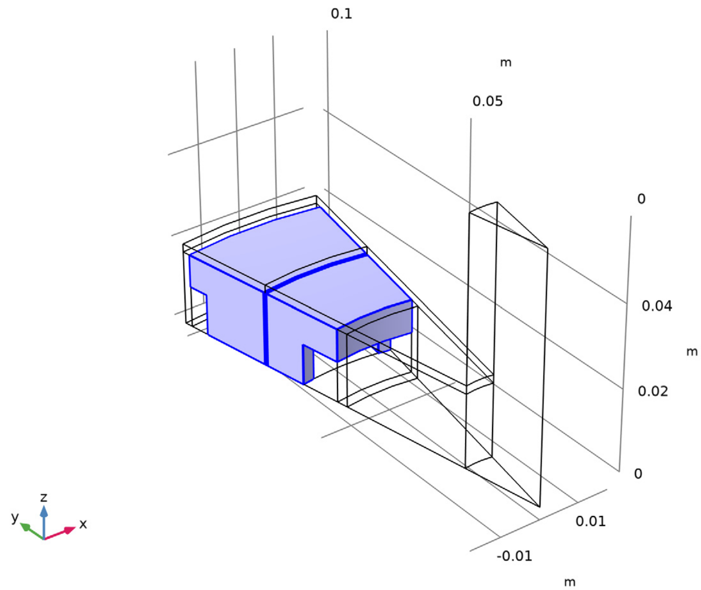
1 In the **Physics** toolbar, click **Domains** and choose **Magnetic Flux Conservation**.

2 In the **Settings** window for **Magnetic Flux Conservation**, type `Magnetic Flux Conservation - air` in the **Label** text field.

3 Locate the **Domain Selection** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `5-6, 10-11` in the **Selection** text field.

5 Click **OK**.



Magnetic Flux Conservation - iron

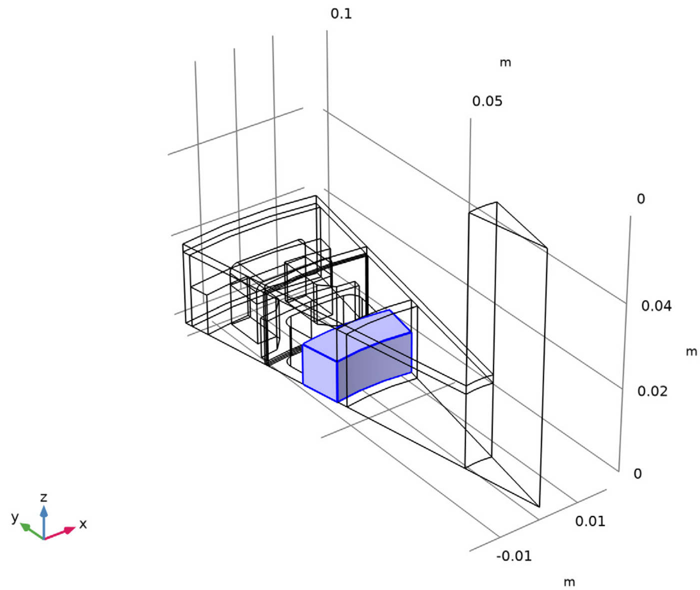
1 In the **Physics** toolbar, click **Domains** and choose **Magnetic Flux Conservation**.

2 In the **Settings** window for **Magnetic Flux Conservation**, type `Magnetic Flux Conservation - iron` in the **Label** text field.

3 Locate the **Domain Selection** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `12` in the **Selection** text field.

5 Click **OK**.



6 In the **Settings** window for **Magnetic Flux Conservation**, locate the **Constitutive Relation B-H** section.

7 From the **Magnetization model** list, choose **B-H curve**.

Ampère's Law - stator core

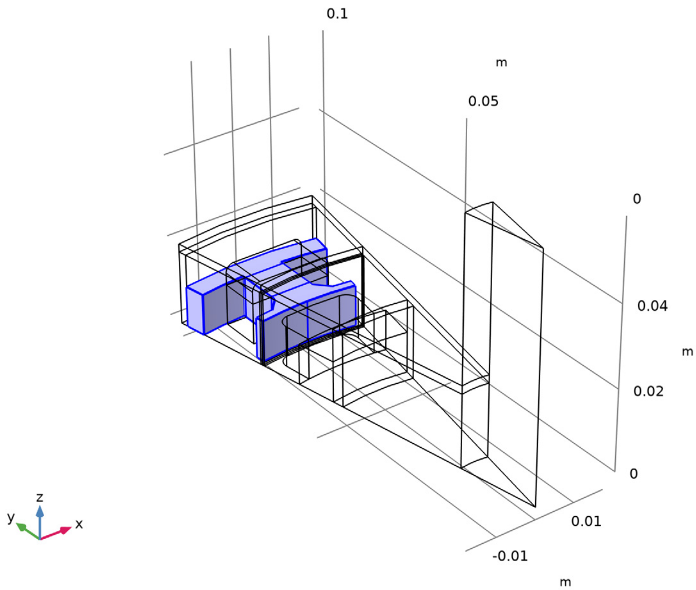
1 In the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 In the **Settings** window for **Ampère's Law**, type `Ampère's Law - stator core` in the **Label** text field.

3 Locate the **Domain Selection** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 4 in the **Selection** text field.

5 Click **OK**.



6 In the **Settings** window for **Ampère's Law**, locate the **Constitutive Relation B-H** section.

7 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.

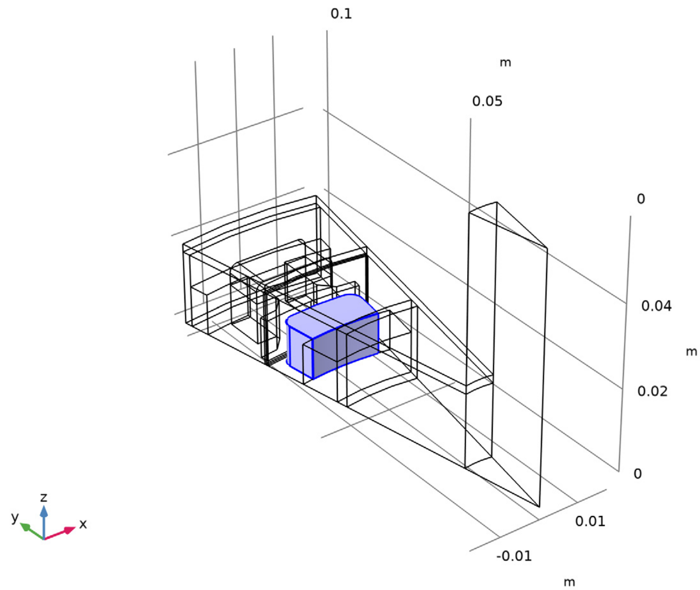
3 From the **Loss model** list, choose **Steinmetz**.

Conducting Magnet 1

Now add a magnet. Leaving the default setting, electrical isolation on all boundaries is assumed. It is possible to change this assumption by changing the constraint for induced currents.

1 In the **Physics** toolbar, click **Domains** and choose **Conducting Magnet**.

2 Select Domain 14 only.



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 100 only.

South 1

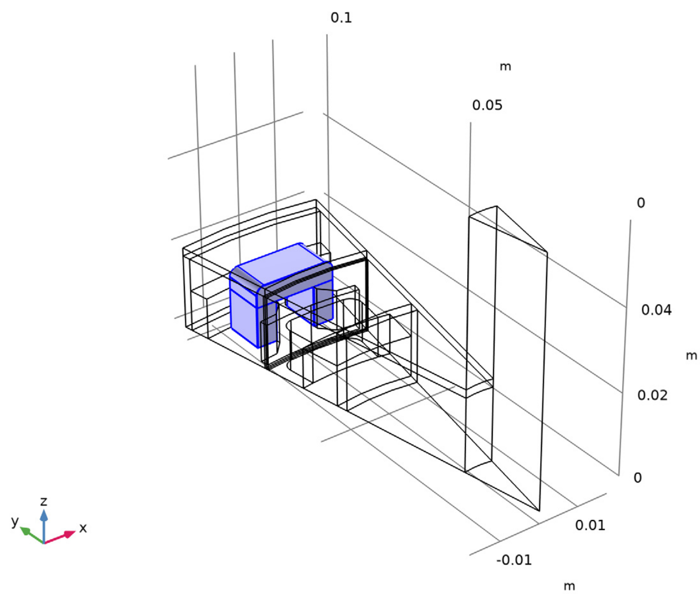
1 In the **Model Builder** window, click **South 1**.

2 Select Boundary 99 only.

Coil 1

1 In the **Physics** toolbar, click **Domains** and choose **Coil**.

2 Select Domains 7–9 only.



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

4 From the **Conductor model** list, choose **Homogenized multiturn**.

5 From the **Coil type** list, choose **Numeric**.

6 In the I_{coil} text field, type `I0*sin(omega*t)`.

7 Locate the **Homogenized Multiturn Conductor** section. In the N text field, type `1`.

8 In the a_{wire} text field, type `a_coil`.

Loss Calculation 1

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

Geometry Analysis 1

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

2 In the **Settings** window for **Geometry Analysis**, locate the **Coil Geometry** section.

3 Find the **Symmetry specification** subsection. In the F_L text field, type `2`.

This accounts for the fact that only one half of the coil is included.

Specify the current direction in the coil.

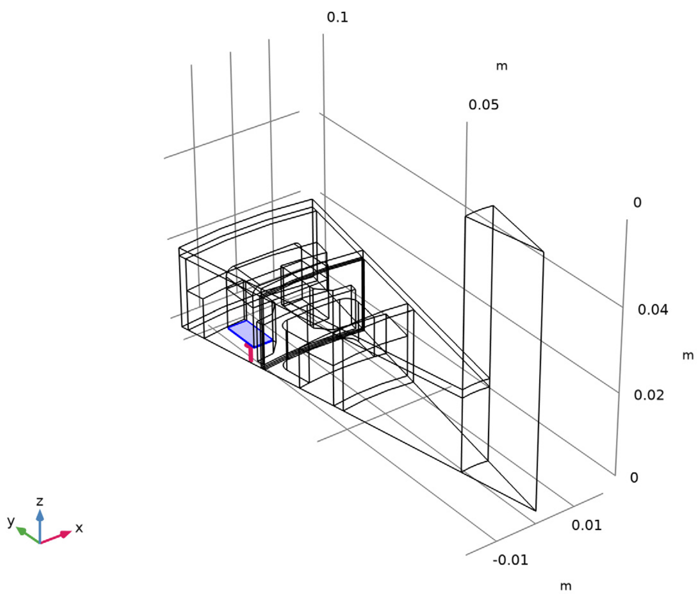
Input 1

1 In the **Model Builder** window, expand the **Geometry Analysis 1** node, then click **Input 1**.

2 Select Boundary 43 only.

Zoom out to see the direction arrow.

3 Click the **Zoom Extents** button in the **Graphics** toolbar.



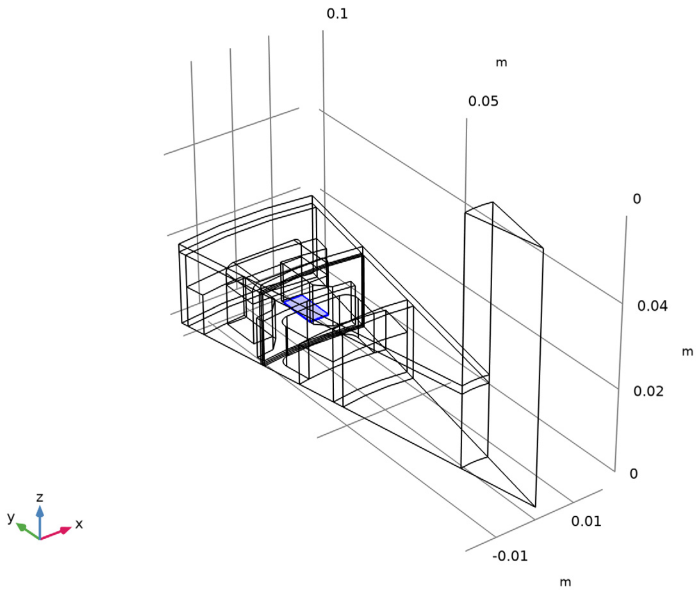
Geometry Analysis 1

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

Output 1

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

2 Select Boundary 61 only.



ROTATING MACHINERY, MAGNETIC (RMM)

Coil 1

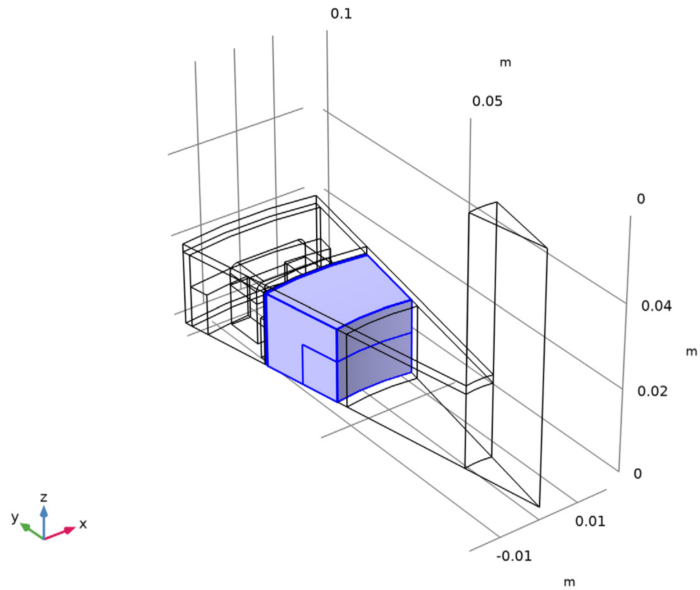
In the **Model Builder** window, collapse the **Component 1 (comp1)>Rotating Machinery, Magnetic (rmm)>Coil 1** node.

COMPONENT 1 (COMP1)

Rotating Domain 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Domains>Rotating Domain**.

2 Select Domains 10–12 and 14 only.



3 In the **Settings** window for **Rotating Domain**, locate the **Rotation** section.

4 In the α text field, type `omega_rotor*t`.

5 Locate the **Axis** section. Specify the $\mathbf{u}_{rot}$ vector as

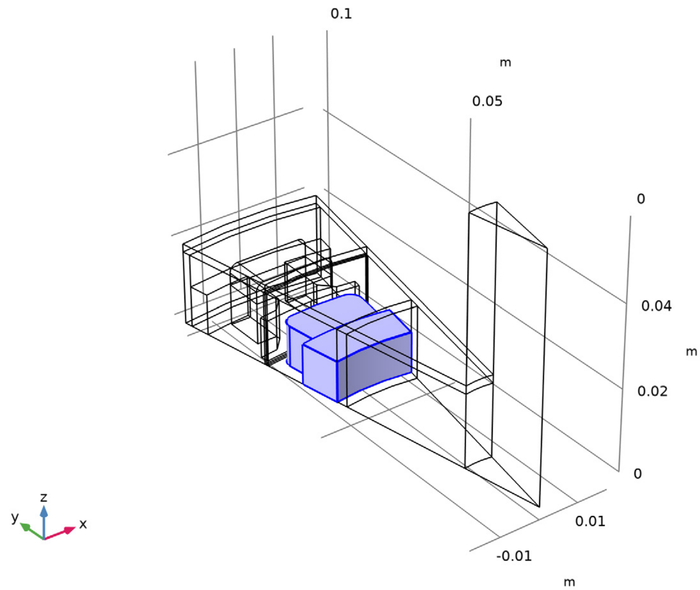
0	X
0	Y
-1	Z

ROTATING MACHINERY, MAGNETIC (RMM)

Force Calculation 1

1 In the **Physics** toolbar, click **Domains** and choose **Force Calculation**.

2 Select Domains 12 and 14 only.



Gauge Fixing for A-field 1

Fix the gauge for the magnetic vector potential.

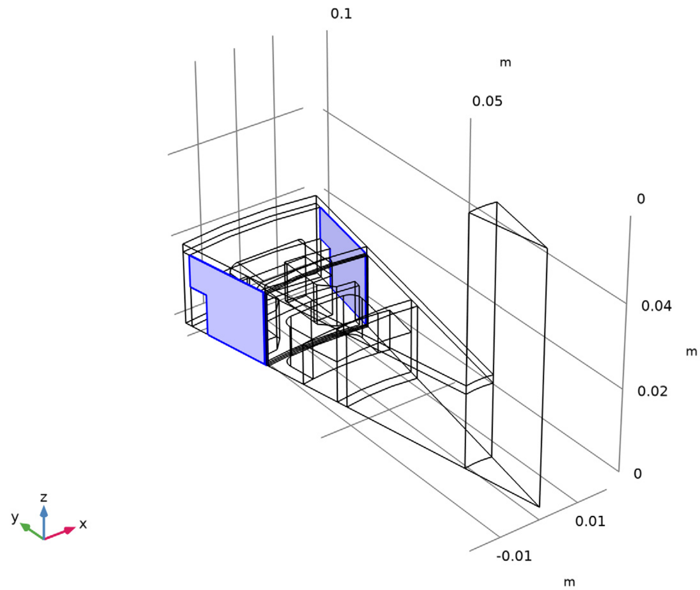
In the **Physics** toolbar, click **Domains** and choose **Gauge Fixing for A-field**.

Periodic Condition 1

Set up the periodicity of the model. Use separate features for the stator and rotor and, for vector and scalar potentials respectively.

1 In the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.

2 Select Boundaries 26, 32, 71, and 72 only.



3 In the **Settings** window for **Periodic Condition**, locate the **Periodic Condition** section.

4 From the **Type of periodicity** list, choose **Antiperiodicity**.

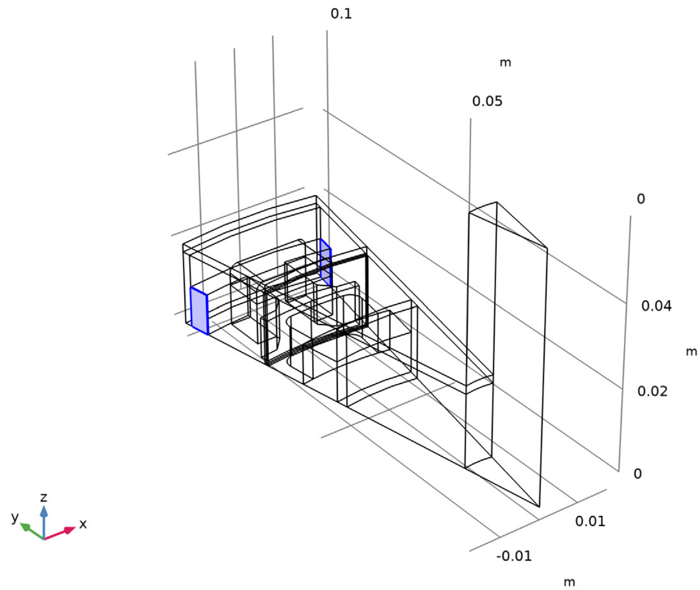
Periodic Condition 2

1 Right-click **Periodic Condition 1** and choose **Duplicate**.

2 In the **Settings** window for **Periodic Condition**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 23 and 73 only.



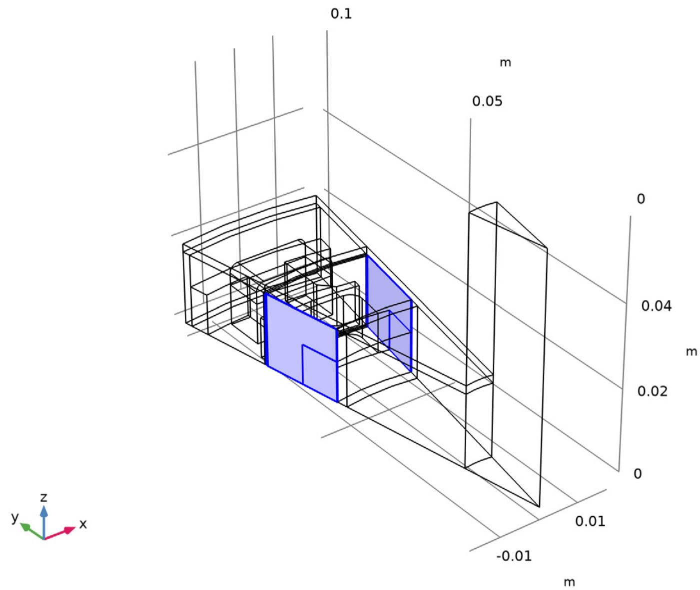
Periodic Condition 3

1 Right-click **Periodic Condition 2** and choose **Duplicate**.

2 In the **Settings** window for **Periodic Condition**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 74, 78, 82, and 113–115 only.



Sector Symmetry 1

Add the pair condition for the rotor-stator interface.

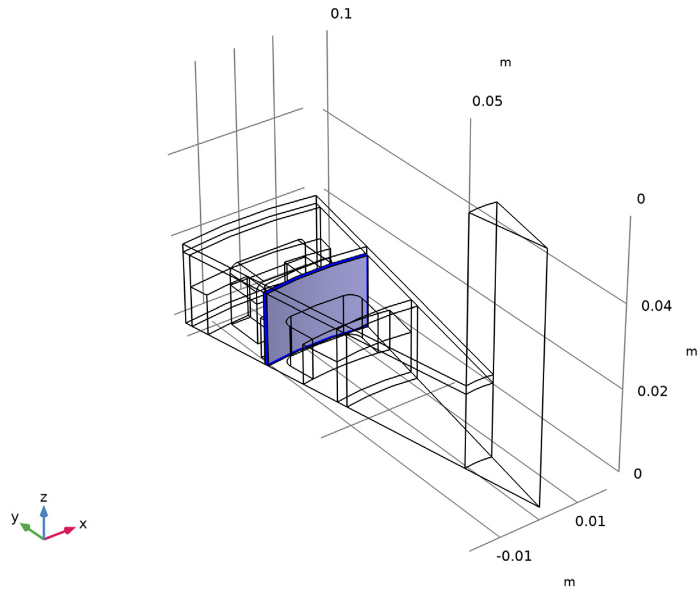
1 In the **Physics** toolbar, click **Pairs** and choose **Sector Symmetry**.

2 In the **Settings** window for **Sector Symmetry**, locate the **Pair Selection** section.

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Identity Boundary Pair 3 (ap3)** in the **Pairs** list.

5 Click **OK**.



6 In the **Settings** window for **Sector Symmetry**, locate the **Sector Settings** section.

7 In the n_{sect} text field, type `n_sectors`.

8 From the **Type of periodicity** list, choose **Antiperiodicity**.

Switch on the option to see more physics options.

9 Click the **Show More Options** button in the **Model Builder** toolbar.

10 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

11 Click **OK**.

Use weak constraints for improved numerical stability.

12 In the **Settings** window for **Sector Symmetry**, click to expand the **Constraint Settings** section.

13 Select the **Use weak constraints** check box.

Arkkio Torque Calculation 1

Add the torque calculation by means of Arkkio's method.

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

DEFINITIONS

Set up variables and other definitions used to define customized output.

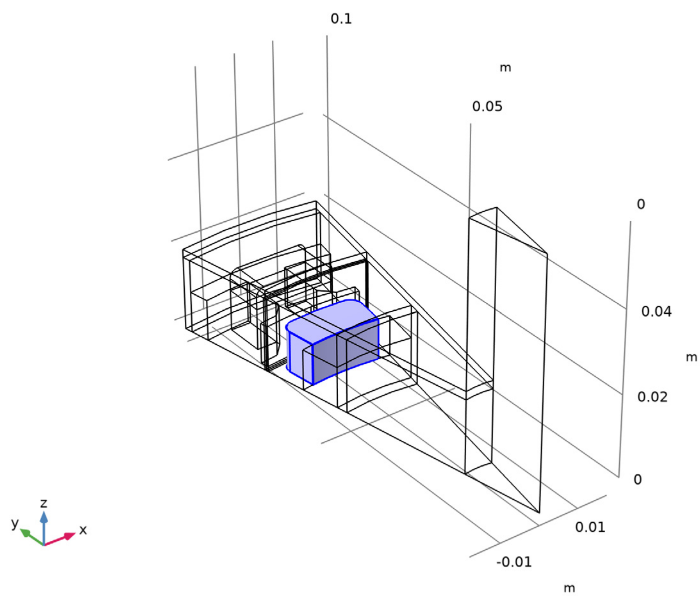
Integration - Magnet

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `Integration - Magnet` in the **Label** text field.

3 In the **Operator name** text field, type `intop1_magnet`.

4 Select Domain 14 only.



Integration - Coil

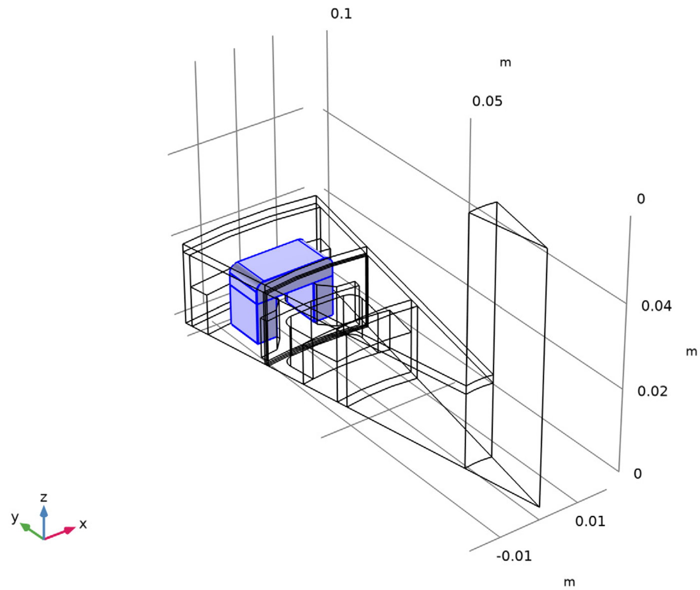
1 Right-click **Integration - Magnet** and choose **Duplicate**.

2 In the **Settings** window for **Integration**, type `Integration - Coil` in the **Label** text field.

3 In the **Operator name** text field, type `intop2_coil`.

4 Locate the **Source Selection** section. Click **Clear Selection**.

5 Select Domains 7–9 only.



Global Variable Probe 1 - Torque

Define probes to be plotted while solving.

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

2 In the **Settings** window for **Global Variable Probe**, type `Global Variable Probe 1 - Torque` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `rmm.Tax_0*n_sectors*2`.

4 Select the **Description** check box. In the associated text field, type `Axial Torque (N*m)`.

Global Variable Probe 2 - Arkkio's Torque method

1 Right-click **Global Variable Probe 1 - Torque** and choose **Duplicate**.

2 In the **Settings** window for **Global Variable Probe**, type `Global Variable Probe 2 - Arkkio's Torque method` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `rmm.Tark_1*2`.

4 In the **Description** text field, type `Arkkio's Torque Method (N*m)`.

Global Variable Probe 3 - Magnet Loss

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

2 In the **Settings** window for **Global Variable Probe**, type `Global Variable Probe 3 - Magnet Loss` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `intop1_magnet(rmm.Qh)*n_sectors*2`.

4 Select the **Description** check box. In the associated text field, type `Losses in Magnets (W)`.

5 Click to expand the **Table and Window Settings** section. From the **Plot window** list, choose **New window**.

Global Variable Probe 4 - Coil Loss

1 Right-click **Global Variable Probe 3 - Magnet Loss** and choose **Duplicate**.

2 In the **Settings** window for **Global Variable Probe**, type `Global Variable Probe 4 - Coil Loss` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `intop2_coil(rmm.Qh)*n_sectors*2`.

4 In the **Description** text field, type `Losses in Coils (W)`.

DEFINITIONS

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

ROTATING MACHINERY, MAGNETIC (RMM)

In the **Model Builder** window, collapse the **Component 1 (comp1)>Rotating Machinery, Magnetic (rmm)** node.

MESH 1

Next, create the mesh. Use the mesh suggested by the physics as a starting point.

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

Size

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.

2 Right-click **Size** and choose **Build Selected**.

Use a finer mesh on the side of the magnet facing the stator.

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 **Boundary**.

4 Select Boundaries 92, 100, and 109 only.

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

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

7 Select the **Maximum element size** check box. In the associated text field, type `0.0005`.

8 Click **Build Selected**.

The pair boundaries need a finer mesh on the destination boundary so custom meshing is needed for source and destination.

Free Triangular 2

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

2 Select Boundary 36 only.

Size 1

1 Right-click **Free Triangular 2** and choose **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.

5 Select the **Maximum element size** check box. In the associated text field, type `0.001`.

Free Triangular 3

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 2** and choose **Duplicate**.

2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 75 only.

Size 1

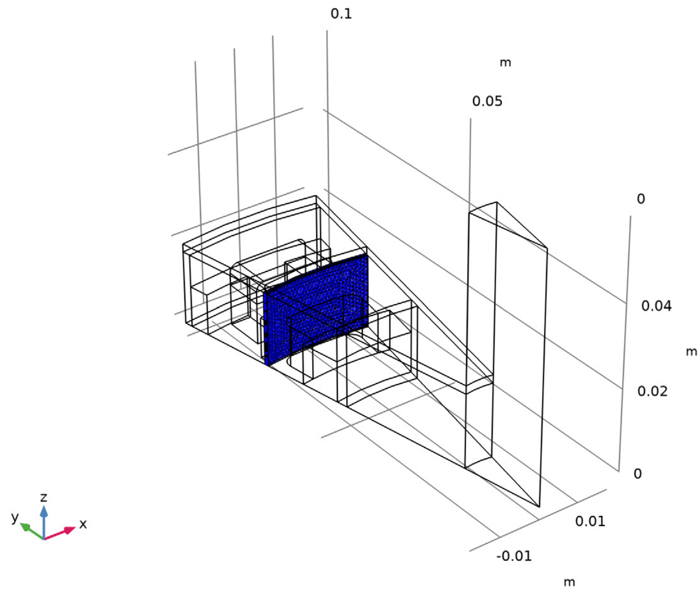
1 In the **Model Builder** window, expand the **Free Triangular 3** node, then click **Size 1**.

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

3 In the **Maximum element size** text field, type `0.001/1.25`.

4 Click **Build Selected**.

5 Click the Zoom Extents button in the Graphics toolbar.



Size 1

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

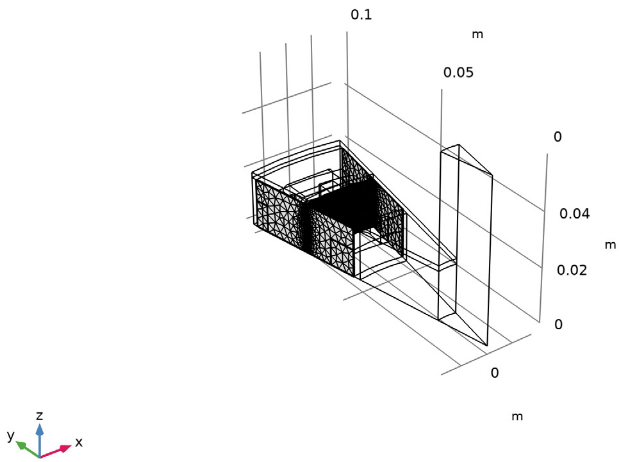
4 Click **Build Selected**.

The periodic boundaries need identical meshes and this part was set up by the physics.

Copy 3

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Copy 3** and choose **Build Selected**.

2 Click the **Zoom Extents** button in the **Graphics** toolbar.



Free Tetrahedral 1

1 In the **Model Builder** window, click **Free Tetrahedral 1**.

2 Select Domains 4–12 only.

Use the free tetrahedral mesh in all domains except the magnet.

Size 1

1 Right-click **Free Tetrahedral 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 **Edge**.

4 Select Edges 105 and 124 only.

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

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

7 Select the **Maximum element size** check box. In the associated text field, type `0.001/2`.

   Use a boundary layer mesh to resolve the skin depth in the magnet.

8 Click **Build Selected**.

Boundary Layers 1

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

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

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

4 Select Domain 14 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 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `91-92, 94, 99-100, 109-110` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

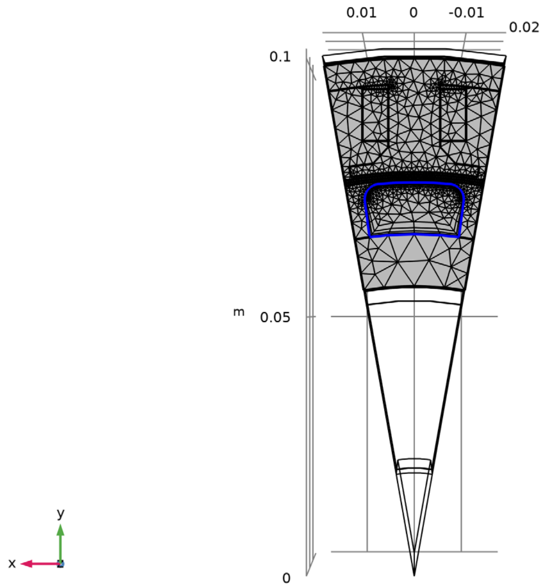
7 In the **Number of layers** text field, type `5`.

8 In the **Stretching factor** text field, type `1.8`.

9 Click **Build Selected**.

10 Click the **Go to XY View** button in the **Graphics** toolbar three times.

11 Click the **Zoom Extents** button in the **Graphics** toolbar.



STUDY 1

Next set up the stationary study that will compute the initial conditions for the time-dependent simulation.

First we need to solve for the numeric coil.

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

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

3 Clear the **Generate default plots** check box.

Coil Geometry Analysis

1 In the **Study** toolbar, click **Study Steps** and choose **Other>Coil Geometry Analysis**.

2 Right-click **Study 1>Step 2: Coil Geometry Analysis** and choose **Move Up**.

Solution 1 (sol1)

The time-dependent step benefits from accurate initial conditions so tighten the tolerance a bit.

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

2 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Stationary Solver 2**.

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1e-6`.

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

RESULTS

Create a custom plot.

Magnetic Flux Density (stationary)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Magnetic Flux Density (stationary)` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

Volume 1

Right-click **Magnetic Flux Density (stationary)** and choose **Volume**.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Magnetic Flux Density (stationary)** and choose **Arrow Surface**.

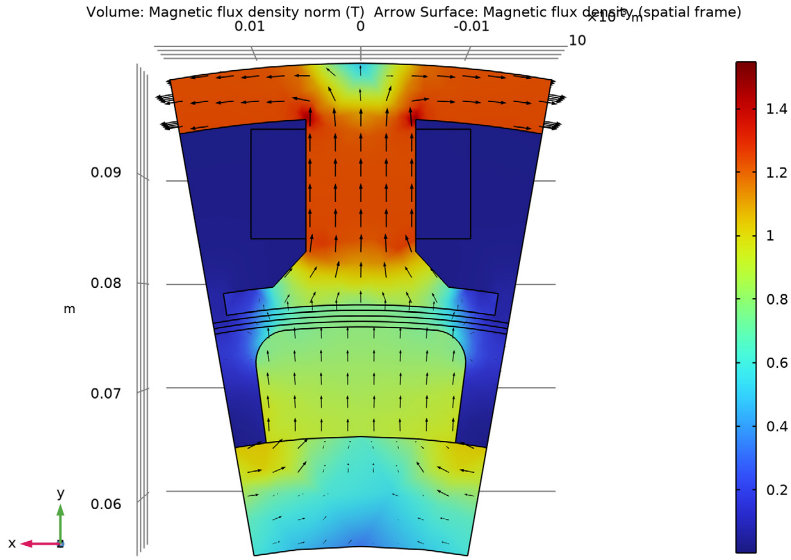
2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Rotating Machinery, Magnetic (Magnetic Fields)>Magnetic>rmm.Bx,...,rmm.Bz - Magnetic flux density (spatial frame)**.

3 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `2000`.

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

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

6 Click the **Zoom Extents** button in the **Graphics** toolbar.



ADD STUDY

Proceed to set up the time-dependent simulation using the stationary solution as initial condition. The latter is necessary as otherwise the permanent magnet would be interpreted as being switched on at $t = 0$, resulting in an unphysical solution.

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> Time Dependent**.

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

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

STUDY 2

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

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

3 Clear the **Generate default plots** check box.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog box, type `time_one_cycle/25` in the **Step** text field.

5 In the **Stop** text field, type `1.5*time_one_cycle`.

6 Click **Replace**.

Proper setup of initial conditions and handling of variables that are not solved for, in this case variables used by the coil geometry analysis, requires some extra attention.

7 In the **Settings** window for **Time Dependent**, click to expand the **Values of Dependent Variables** section.

8 Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

9 From the **Method** list, choose **Solution**.

10 From the **Study** list, choose **Study 1, Stationary**.

11 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

12 From the **Method** list, choose **Solution**.

13 From the **Study** list, choose **Study 1, Stationary**.

These steps make sure that we get the desired initial values and use and output the desired values of whatever variables we are not solving for in the current study step.

Similar to what we did for the mesh, we initially let COMSOL set up the solver for us but before solving we make some important customization.

Solution 3 (sol3)

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

2 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 3 (sol3)** click **Dependent Variables 1**.

3 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.

4 From the **Method** list, choose **None**.

5 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 3 (sol3)** click **Time-Dependent Solver 1**.

6 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

In order to get highly accurate output for the time derivatives (electric field and eddy current losses), the solver has been forced to hit the output times (steps taken by solver: **Strict**).

Limiting the maximum BDF order to 2 is a safeguard when handling systems that potentially have oscillatory solutions.

Algebraic variables, that is variables that have no direct rate dependency, can cause slow and sluggish time stepping. In this model, the Lagrange multipliers for the weak constraints in the sector symmetry feature and the additional variable introduced by the gauge fixing feature are algebraic. Circuit couplings may also introduce algebraic variables. Excluding algebraic states from the error estimation will speed up the solution (error estimation: **Exclude algebraic**).

7 Click to collapse the **Time Stepping** section. The PARDISO direct solver is usually a bit faster and leaner on memory than the default direct solver (MUMPS) on this type of model.

8 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 3 (sol3)>Time-Dependent Solver 1** node, then click **Direct**.

9 In the **Settings** window for **Direct**, locate the **General** section.

10 From the **Solver** list, choose **PARDISO**.

Any time-dependent modeling involving moving meshes should make use of frequent Jacobian updates. Also when there are nonlinear materials, allowing for more iterations and solving the stationary nonlinearity to a higher accuracy is recommended.

11 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 3 (sol3)>Time-Dependent Solver 1** click **Fully Coupled 1**.

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

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

14 In the **Maximum number of iterations** text field, type `8`.

15 In the **Tolerance factor** text field, type `1e-3`.

RESULTS

The setup of the time-dependent solver is almost finished but for models that take some considerable time to simulate, it is good practice to generate some customized graphical output while solving for debugging purposes.

Study 2/Solution 3 (sol3)
A dataset for the time-dependent solution was generated with the solver. Use this to create the desired plot to be shown while solving.

First, change the frame to plot in the observer's frame (spatial).

Current Density, Magnet (transient)
Now, proceed to add the plot.

1 In the **Model Builder** window, expand the **Results>Datasets** node.
2 Right-click **Results** and choose **3D Plot Group**.
3 In the **Settings** window for **3D Plot Group**, type `Current Density, Magnet (transient)` in the **Label** text field.
4 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.
5 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Domain**.
6 Select Domain 14 only.

Volume 1
1 Right-click **Current Density, Magnet (transient)** and choose **Volume**.
2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Rotating Machinery, Magnetic (Magnetic Fields)>Currents and charge>rmm.normJ - Current density norm - A/m²**.

Plot the current density magnitude in the magnet only.

Arrow Surface 1
1 In the **Model Builder** window, right-click **Current Density, Magnet (transient)** and choose **Arrow Surface**.
2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Rotating Machinery, Magnetic (Magnetic Fields)>Currents and charge>rmm.Jx,...,rmm.Jz - Current density (spatial frame)**.
3 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `400`.
4 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

Current Density, Magnet (transient)
Make sure that the geometry outline is following the motion.

1 In the **Model Builder** window, click **Current Density, Magnet (transient)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **Frame** list, choose **Spatial (x, y, z)**.

STUDY 2

Finally activate the plotting during solution of the newly created plot group.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Current Density, Magnet (transient)**.

Note that also the probes will be plotted at the internal step rate of the solver which is usually higher than the solution output rate.

Now, it is time to solve the model. This will take of the order of one hour - more or less depending on computer hardware.

5 In the **Home** toolbar, click = **Compute**.

RESULTS

Torque

Inspect the probes, start with the torque plot.

Activate legends to distinguish between the curves.

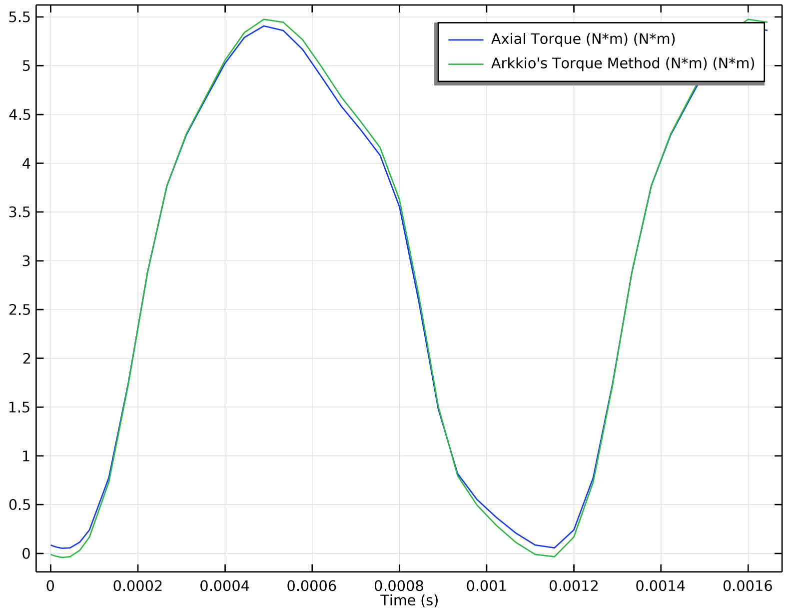
1 In the **Model Builder** window, under **Results** click **Probe Plot Group 1**.

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

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Torque** node, then click **Probe Table Graph 1**.

2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.



There is good agreement between Arkkio's torque and the torque computed using the Maxwell stress tensor.

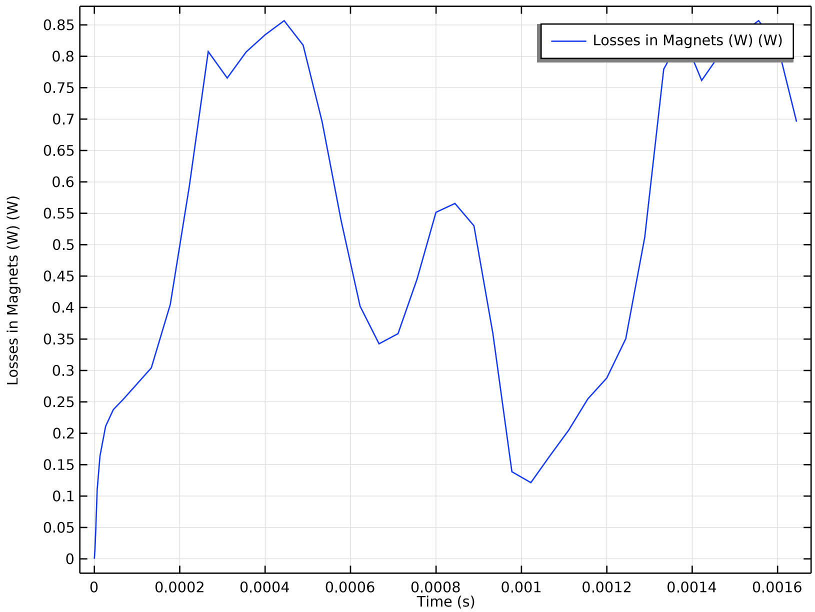
Next, have a look at the eddy current losses in the magnet.

Losses in Magnets

1 In the **Model Builder** window, expand the **Results>Probe Plot Group 2** node, then click **Probe Plot Group 2**.

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

Probe Table Graph 1



The magnet losses vary significantly over time.

Finally have a look at the losses in the coil.

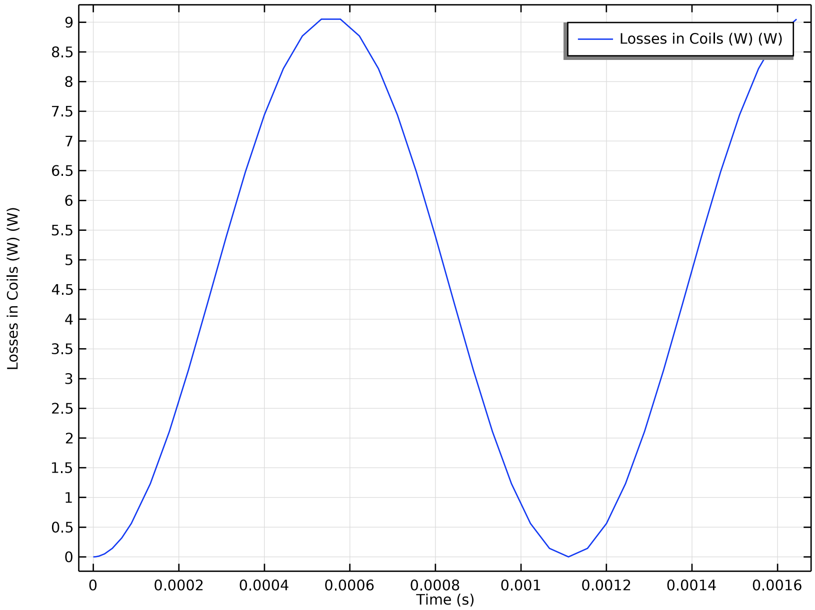
Losses in Coils

1 In the **Model Builder** window, expand the **Results>Probe Plot Group 3** node, then click **Probe Plot Group 3**.

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

Probe Table Graph 1

The coil has a prescribed sinusoidal current density giving rise to resistive losses in the copper.



Next, proceed to create custom plots.

Study 2/Solution 3 (5) (sol3)

In the **Model Builder** window, under **Results>Datasets** right-click **Study 2/Solution 3 (sol3)** and choose **Duplicate**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Select Domains 4, 12, and 14 only.

Datasets with selections can be used as an alternative to adding selections directly on the plot features.

Next, add a plot of the magnetic flux density.

Magnetic Flux Density (transient)

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Model Builder** window, click **3D Plot Group 6**.

3 In the **Settings** window for **3D Plot Group**, type `Magnetic Flux Density (transient)` in the **Label** text field.

4 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 3 (5) (sol3)**.

5 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

Volume 1

Right-click **Magnetic Flux Density (transient)** and choose **Volume**.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Magnetic Flux Density (transient)** and choose **Arrow Surface**.

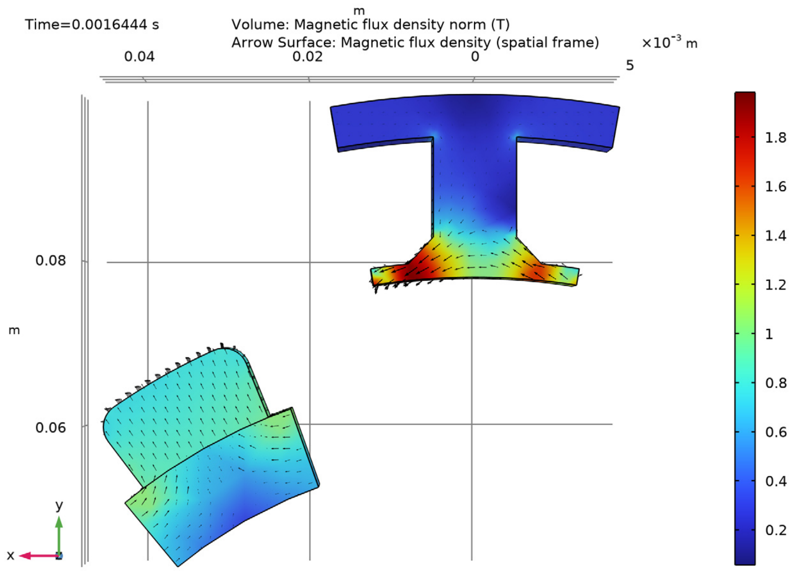
2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Rotating Machinery, Magnetic (Magnetic Fields)>Magnetic>rmm.Bx,...,rmm.Bz - Magnetic flux density (spatial frame)**.

3 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `2000`.

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

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

6 Click the **Zoom Extents** button in the **Graphics** toolbar.



The magnetic flux density in the magnet and iron is shown.

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.

LOSS CALCULATION

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

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

3 Clear the **Generate default plots** check box.

4 In the **Label** text field, type `Loss Calculation`.

Step 1: Time to Frequency Losses

1 In the **Model Builder** window, under **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, Time Dependent**.

4 In the **Electrical period** text field, type `time_one_cycle`.

5 In the **Number of harmonics** text field, type `12`.

6 In the **Home** toolbar, click **Compute**.

RESULTS

Loss Density in Magnets

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Loss Density in Magnets` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Loss Calculation/Solution 4 (sol4)**.

Volume 1

1 Right-click **Loss Density in Magnets** and choose **Volume**.

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

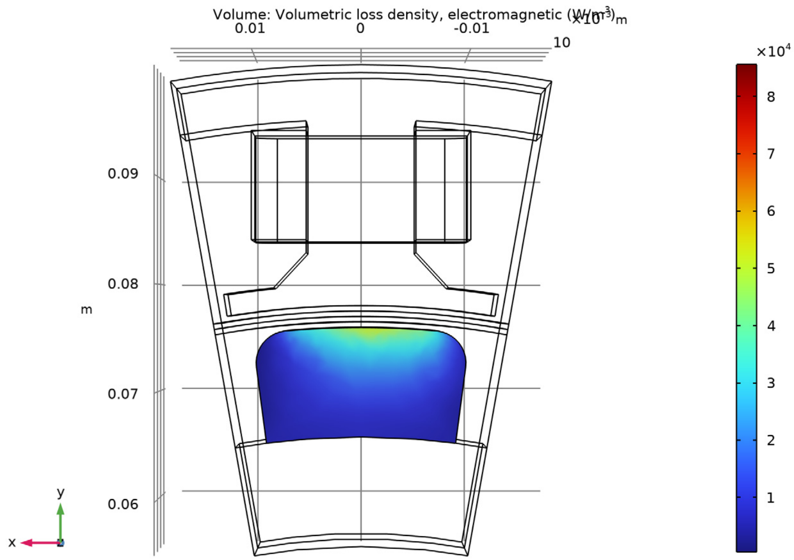
3 In the **Expression** text field, type `rmm.Qh`.

Selection 1

1 Right-click **Volume 1** and choose **Selection**.

2 Select Domain 14 only.

3 In the **Loss Density in Magnets** toolbar, click **Plot**.



Volume Integration 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Integration> Volume Integration**.

2 In the **Settings** window for **Volume Integration**, locate the **Data** section.

3 From the **Dataset** list, choose **Loss Calculation/Solution 4 (sol4)**.

4 Select Domain 14 only.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
rmm.Qh*n_sectors*2	W	Total loss power of the magnet

6 Click **Evaluate**.

The total loss power of the magnet is about 0.85 W.

ROOT

Finally add a suitable thumbnail to the model.

1 In the **Model Builder** window, click the root node.

2 In the root node's **Settings** window, locate the **Presentation** section.

3 Find the **Thumbnail** subsection. Click **Set from Graphics Window**.