Created in COMSOL Multiphysics 6.2

Modeling of an Electric Generator in 3D

Introduction

This application illustrates how to model an electric machine, such as a generator, motor, or drive, by exploiting its sector symmetry to reduce the size of the problem. The machine being studied is a simplified electric generator in 3D, based on the geometry used in the Generator in 2D application.

The application uses the **Rotating Machinery, Magnetic** interface. It is recommended to have a look at the model Rotating Machinery 3D Tutorial before proceeding with this application.

Model Definition

The complete geometry is represented in Figure 1. Parts of the stator have been removed to show the rotor. The core of the rotor and the stator consists of laminated (σ = 0 S/m), saturable iron, and the teeth of the rotor are permanent magnets using available materials.

The generator rotates with a rotational velocity of 60 rpm, and the model is solved in the time domain from t = 0 s to t = 0.25 s, after which the rotor arrives to a configuration symmetrically equivalent to the starting one.

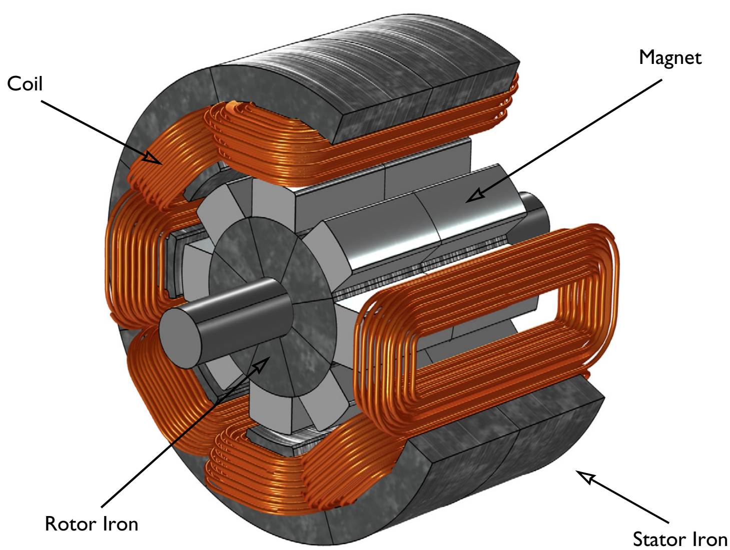


Figure 1: Geometry of the generator.

The geometry has two types of symmetry: the sector symmetry and a reflection symmetry with respect to the midplane orthogonal to the axis, so it can be reduced to the sector geometry shown in Figure 2. The figure indicates the appropriate conditions to use on the symmetry cut boundaries:

- **Periodic Conditions** must be used on the sides of the sector. The type of periodicity chosen is **Antiperiodicity**, since the inputs to the model (the remanent flux density in the permanent magnets) change sign in adjacent sectors.
- The **Sector Symmetry** pair condition is applied on the identity pair created by the geometry at the contact boundary between rotor and stator. The type of periodicity must match the type specified in the Periodic Boundary Condition features, that is, **Antiperiodicity**.
- The reflection symmetry forces the normal component of the magnetic field to be zero at the midplane. This condition is imposed by the default **Magnetic Insulation** feature.

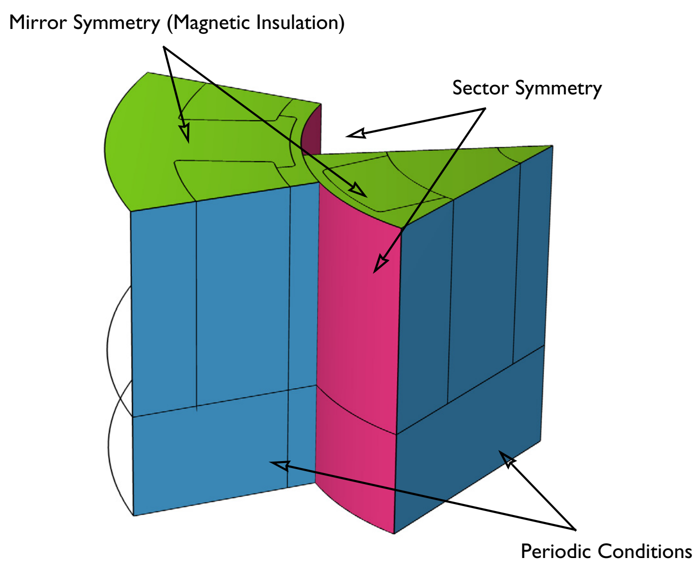


Figure 2: Boundary conditions used to take into account the model symmetry.

MODELING THE COIL

The stator coil has a nonregular shape, that does not fall in either of the **Linear** or **Circular** categories. The direction of the coil can be computed in a preprocessing **Coil Geometry Analysis** step.

The stator coil is affected by the symmetry cut as well. The modeled length of the coil is 1/16 of its actual length (due to the 8-fold sector symmetry and the mirror symmetry). To ensure that the lumped quantities such as the coil voltage or the current are computed correctly, specify appropriate **Symmetry factors** in the **Geometry Analysis** subnode under the **Coil** node.

THE MIXED FORMULATION

The application uses the mixed formulation functionality of the **Rotating Machinery, Magnetic** interface. It solves for the magnetic scalar potential V_{m} in nonconductive regions and for the magnetic vector potential $\mathbf{A}$ in conductive regions. The scalar formulation is solved by **Magnetic Flux Conservation** features, while the vector formulation is solved by **Ampère's Law** features. Advantages of using the scalar potential formulation in the air gap and nonconductive regions are:

- More accurate magnetic flux conservation by the pair coupling (Sector Symmetry). For this reason, it is important to use the scalar potential in the regions adjacent to a pair.
- Decreased number of degrees of freedom compared with the vector formulation.

The limitation of the scalar formulation is that it cannot be used in conductive domains or domains carrying currents, nor in regions that contain closed loops of current (for example, it cannot be used in the air region surrounding the coil).

Care must be taken when using a periodic condition in a sector-symmetric model, since the topological condition on the scalar potential regions must be fulfilled in the complete geometry as well.

DIRECT SOLVERS AND UNIQUENESS OF THE SOLUTION

The **Rotating Machinery, Magnetic** interface, by default, uses a direct solver for stationary and transient simulations. Direct solvers are typically better performing than iterative solvers at the cost of increased memory usage.

Direct solvers can only find a solution if it is unique, unlike iterative solvers. The vector potential formulation is subject to gauge freedom, meaning that the solution is unique up to a gauge transformation. To ensure a globally unique solution for the stationary study step, it is necessary to choose (fix) the gauge by using the **Gauge Fixing for A-Field** feature in all domains where the vector formulation is used. For the time-dependent study step, however, the gauge fixing is not required if the conductivity of these domains is not zero in the numerical sense. More information about the electromagnetic gauge and gauge fixing can be found in the *AC/DC Module User's Guide*.

LOSS CALCULATION

The resistive and iron losses are computed with the use of a **Loss Calculation** subfeature and a **Time to Frequency Losses** study. For the iron part of the generator, the Bertotti loss model is selected to compute the cycle averaged loss power density. A volume integration is then made to compute the total loss power of the generator. The generated voltage and the losses are compared with the 2D counterpart of this model, that is, Generator in 2D.

Results and Discussion

A sector plot of magnetic flux density is presented in Figure 3. The solution is plotted in the spatial frame, in which the rotor moves.

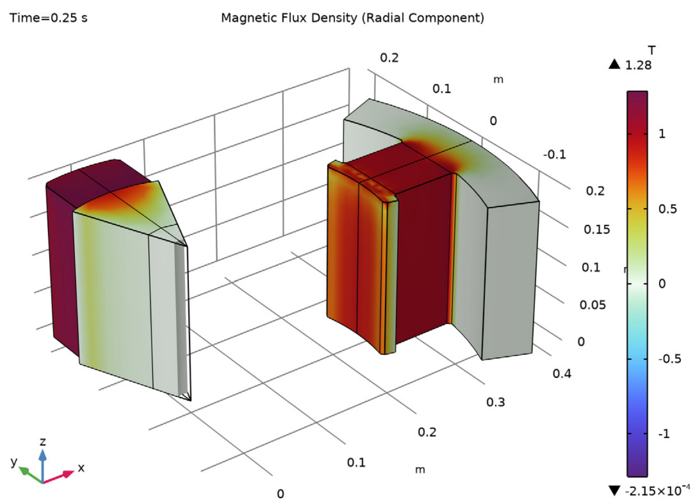


Figure 3: Sector plot of the magnetic flux density.

To visualize the solution in the complete geometry, reconstruct it using **Sector 3D** and **Mirror** datasets. These specialized datasets create another solution by rotating and mirroring other datasets according to the specification.

To properly account for the antisymmetry, select the **Invert phase when rotating** check box in the **Sector 3D** dataset. This functionality changes the sign of the solution when creating adjacent sectors. The resulting plot is shown in Figure 4.

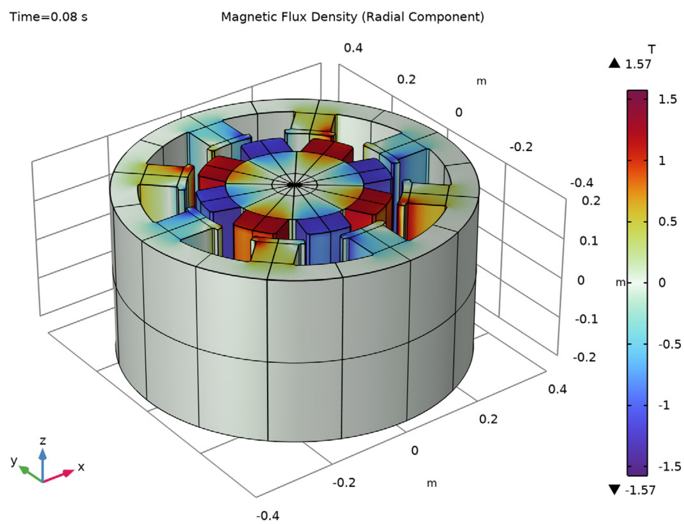


Figure 4: Plot of the magnetic flux density in the complete, reconstructed geometry.

Finally, Figure 5 shows the induced coil voltage as a function of time. The voltage takes into account the number of wires in the multiturn coil and the symmetry. The simulated voltage is in good agreement with the result from the corresponding 2D model, see Figure 3 in the application Generator in 2D.

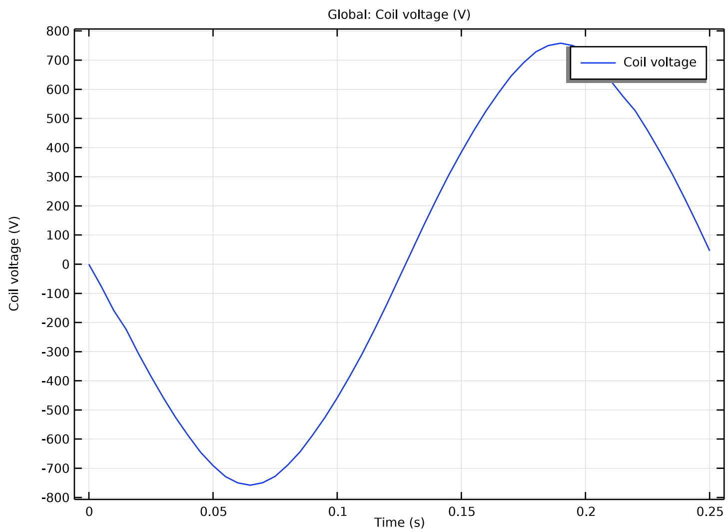


Figure 5: Voltage induced in the complete coil as a function of time.

Application Library path: `ACDC_Module/Devices,_Motors_and_Generators/sector_generator_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 **Preset Studies for Selected Physics Interfaces> Coil Geometry Analysis**.

6 Click **Done**.

GLOBAL DEFINITIONS

Define model parameters: global, constant expressions that can be used anywhere in the application.

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
d_wire	3[mm]	0.003 m	Diameter of wire in the winding
N	100	100	Number of turns in the winding
rpm	60[rpm]	1 1/s	Angular velocity of the rotor

GEOMETRY 1

Insert the geometry sequence from a separate file using the **Insert Sequence** functionality. This functionality copies all the subnodes of the geometry node in the chosen file to the current model.

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `sector_generator_3d_geom_sequence.mph`.

3 In the **Geometry** toolbar, click **Build All**.

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

The geometry represents a sector of the complete generator, further halved along the *xy*-symmetry plane. The geometry is composed of two objects, one for the rotor and one for the stator. They have been constructed from the individual components using two **Union** operations. The geometry is finalized by using **Form Assembly**, which detects the touching boundaries and creates an **Identity Pair** connecting them.

DEFINITIONS

Proceed to the definition of **Selection** nodes: named collection of geometric entities (such as domains or boundaries) that can be reused when applying equations, boundary conditions, and other features. New selections can also be obtained by applying more advanced operations on other selections, such as taking the complement or the adjacent entities. Using selections simplifies the workflow, especially in models with complex geometries. **Explicit** selections are named selections in which the entities (domains, in this case) are selected explicitly.

Cylindrical System 2 (sys2)

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

The cylindrical coordinate system just created will be used to plot the radial flux density in postprocessing.

Stator Coil

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

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

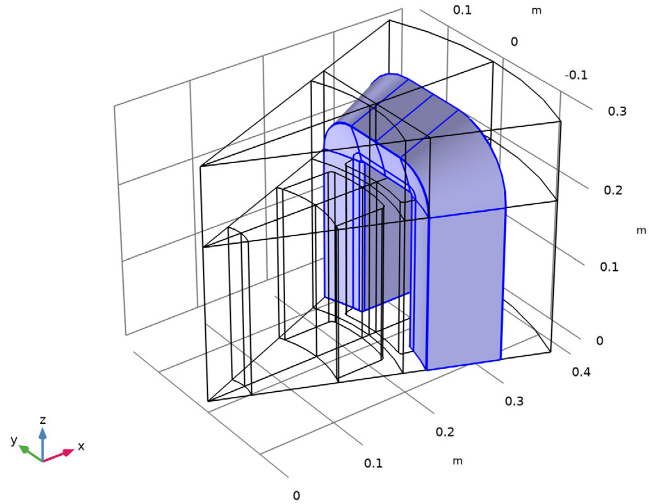
3 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

You can conveniently add domain entities with the **Paste Selection** view instead of clicking the domains manually:

4 Locate the **Input Entities** section. Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `17 19 20 22 23 24` in the **Selection** text field (the domains belonging to the coil).

6 Click **OK**.

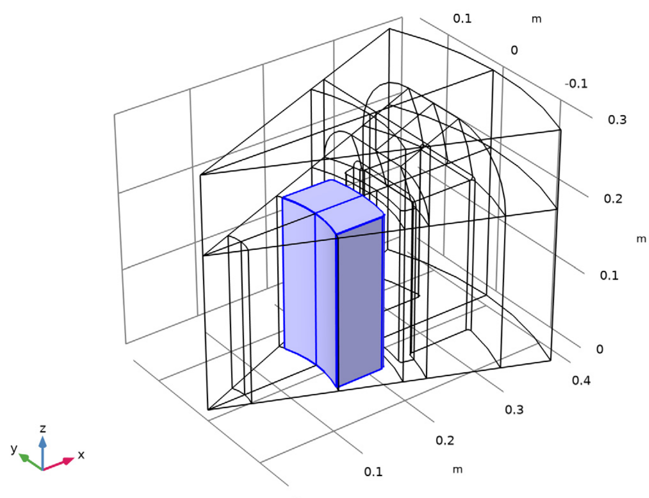


Permanent Magnet

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

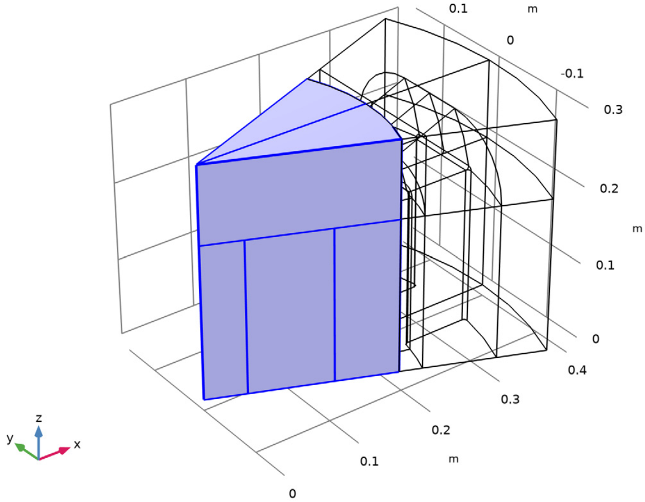
2 In the **Settings** window for **Explicit**, type `Permanent Magnet` in the **Label** text field.

3 Select Domains 9 and 10 only (the magnet domain).



Rotating Domains

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Rotating Domains` in the **Label** text field.
3 Select Domains 1–10 only (all the domains in the rotor).



DEFINITIONS

Create a selection for the stationary domains by taking the complement of the rotor selection.

Stationary Domains

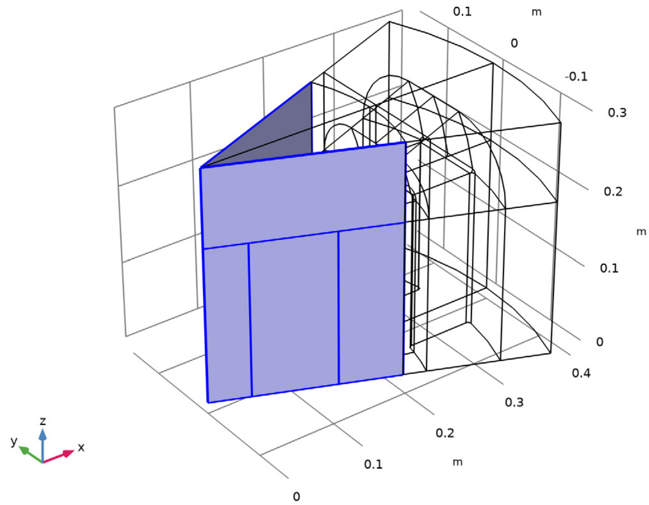
1 In the **Definitions** toolbar, click **Complement**.
2 In the **Settings** window for **Complement**, type `Stationary Domains` in the **Label** text field.
3 Locate the **Input Entities** section. Under **Selections to invert**, click + **Add**.
4 In the **Add** dialog box, select **Rotating Domains** in the **Selections to invert** list.
5 Click **OK**.

Periodic Condition: Rotor

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Periodic Condition: Rotor` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 1, 2, 6, 7, 14, 18, 23, and 27 only.



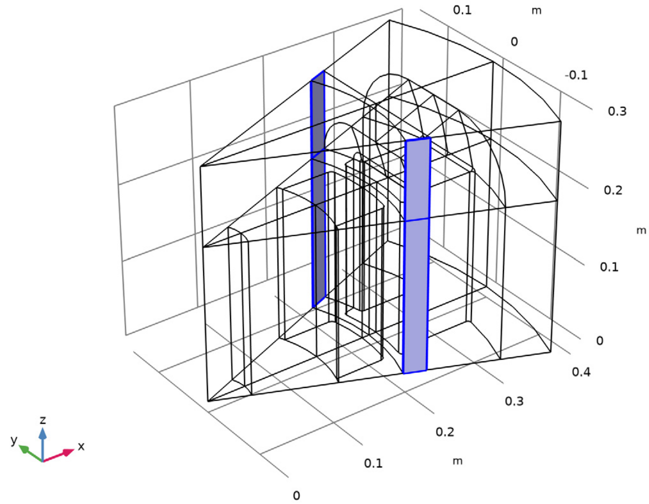
Periodic Condition: Stator, Scalar Potential

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

2 In the **Settings** window for **Explicit**, type `Periodic Condition: Stator, Scalar Potential` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 49, 52, 56, and 59 only.



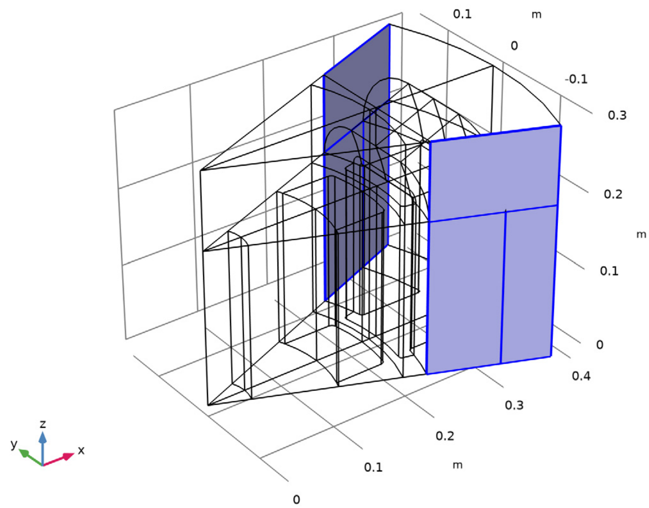
Periodic Condition: Stator, Vector Potential

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

2 In the **Settings** window for **Explicit**, type `Periodic Condition: Stator, Vector Potential` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 75, 78, 96, 98, 118, and 125 only.



Destination

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

2 In the **Settings** window for **Explicit**, type `Destination` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 41–44 only.

Source

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

2 In the **Settings** window for **Explicit**, type `Source` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 48, 51, 55, and 58 only.

Identity Boundary Pair 1 (ap1)

Finally, ensure that the **Destination** side of the identity pair is on the moving domain. This gives a better performance during the solution.

Check the **Source Boundaries** and the **Destination Boundaries** and make sure that the moving domains are in the destination side. The boundaries are numbered progressively with increasing x-coordinate, so the boundaries on the rotor side have lower numbers. Click the **Swap Source and Destination** button to swap the boundary assignment and put the boundaries with the lower numbers in the destination side.

ADD MATERIAL

Proceed with the definition of the materials.

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 Right-click and choose **Add to Component 1 (comp1)**.

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

6 Right-click and choose **Add to Component 1 (comp1)**.

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.

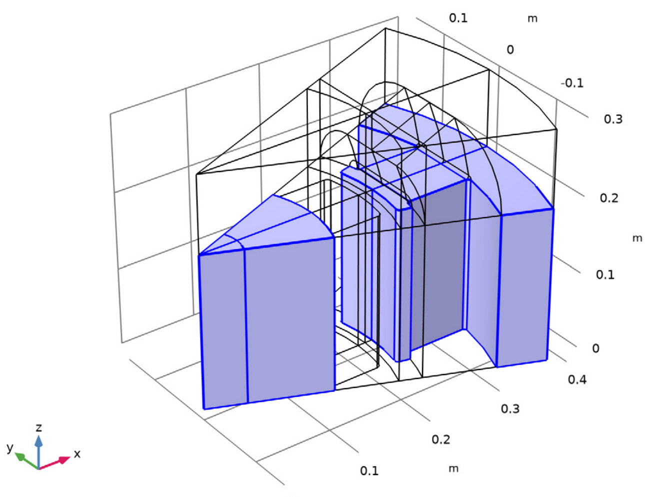
9 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Soft Iron (Without Losses) (mat2)

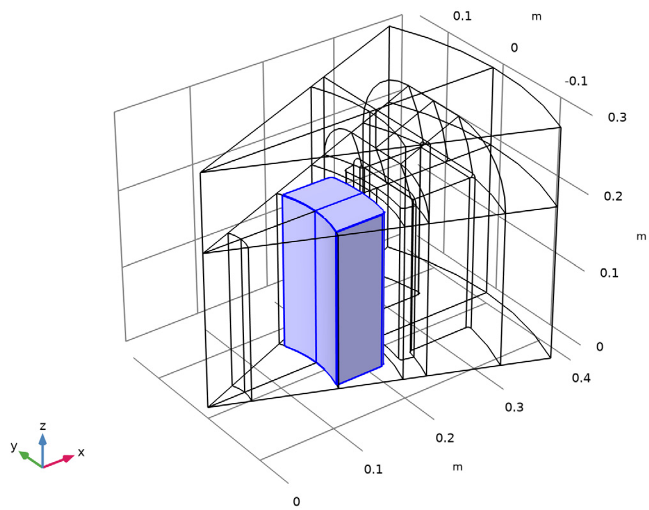
1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Soft Iron (Without Losses) (mat2)**.

2 Select Domains 1, 2, 5, 6, 15, and 16 only.



N50 (Sintered NdFeB) (mat3)

1 In the **Model Builder** window, click **N50 (Sintered NdFeB) (mat3)**.
2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3 From the **Selection** list, choose **Permanent Magnet**.



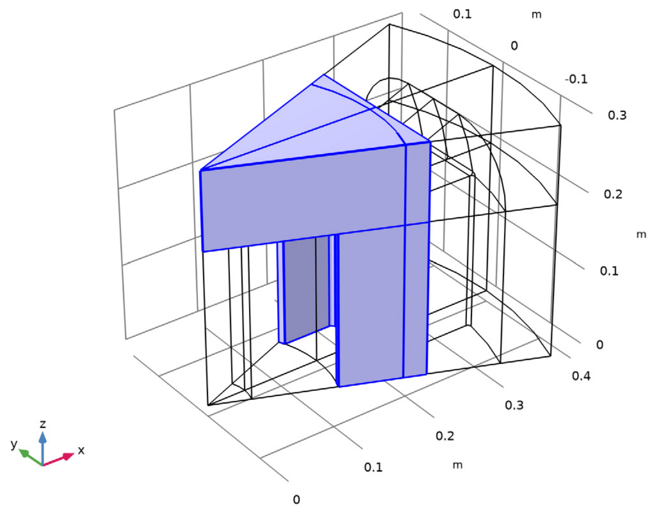
ROTATING MACHINERY, MAGNETIC (RMM)

Move on to the setup of the physics interface.

Magnetic Flux Conservation: Air Gap

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Rotating Machinery, Magnetic (rmm)** and choose **Magnetic Flux Conservation**.
2 In the **Settings** window for **Magnetic Flux Conservation**, type `Magnetic Flux Conservation: Air Gap` in the **Label** text field.

3 Select Domains 3, 4, 7, 8, and 11–14 only.

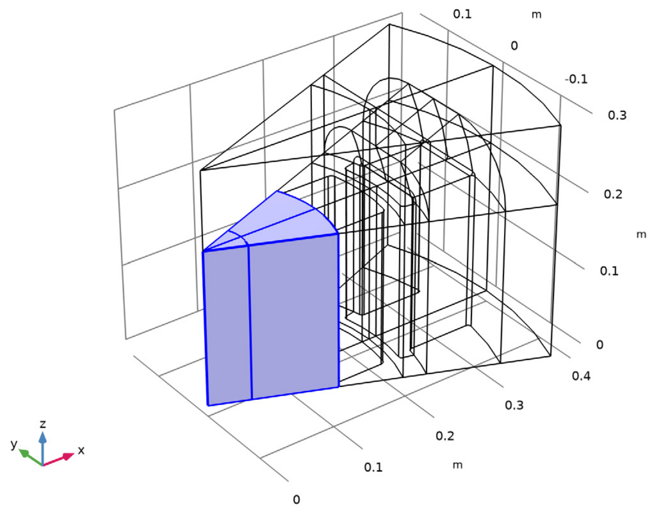


Magnetic Flux Conservation: Rotor Iron

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: Rotor Iron` in the **Label** text field.

3 Select Domains 1, 2, 5, and 6 only.



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**.

Set loss model to **Bertotti**.

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

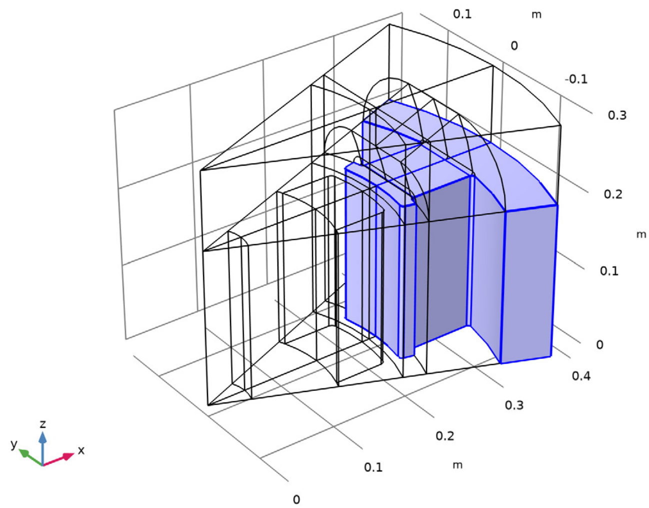
3 From the **Loss model** list, choose **Bertotti**.

Ampère's Law: Stator Iron

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 Iron` in the **Label** text field.

3 Select Domains 15 and 16 only.



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 **Bertotti**.

Add a **Conducting Magnet** feature to define remanent flux of the permanent magnet. Set no constraint for induced currents to obtain similar results as for the 2D counterpart of this model.

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 **Permanent Magnet**.

4 Locate the **Constitutive Relation Jc-E** section. From the **Constrain for induced currents** list, choose **No induced currents constrain**.

North 1

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

2 Select Boundaries 45 and 46 only.

South 1

1. In the **Model Builder** window, click **South 1**.
2. Select Boundaries 30 and 34 only.

Conducting Magnet 1

In the **Model Builder** window, click **Conducting Magnet 1**.

Loss Calculation 1

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

Coil 1

1. In the **Physics** toolbar, click **Domains** and choose **Coil**.
2. In the **Settings** window for **Coil**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Stator Coil**.
4. Locate the **Coil** section. From the **Conductor model** list, choose **Homogenized multiturn**.
5. From the **Coil type** list, choose **Numeric**.

   This setting requires solving a **Coil Geometry Analysis** preprocessing step.
6. In the I_{coil} text field, type `0[A]`.
7. Locate the **Homogenized Multiturn Conductor** section. In the N text field, type `N`.
8. From the **Coil wire cross-section area** list, choose **From round wire diameter**.
9. In the d_{wire} text field, type `d_wire`.

Geometry Analysis 1

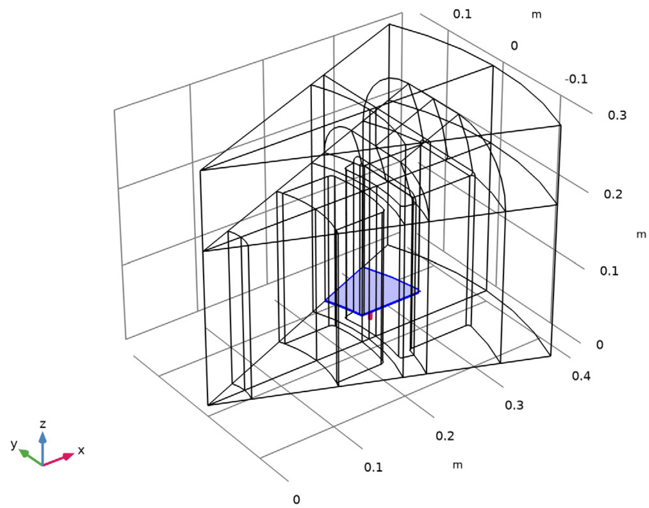
The coil domain is only 1/16 of the total length of the coil (due to symmetry). Use the functionality in the **Geometry Analysis** subfeature to apply the appropriate corrections to the lumped quantities, such as the induced voltage.

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 `16`.

Input 1

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

2 Select Boundary 97 only (one of the two boundaries at the symmetry cut).



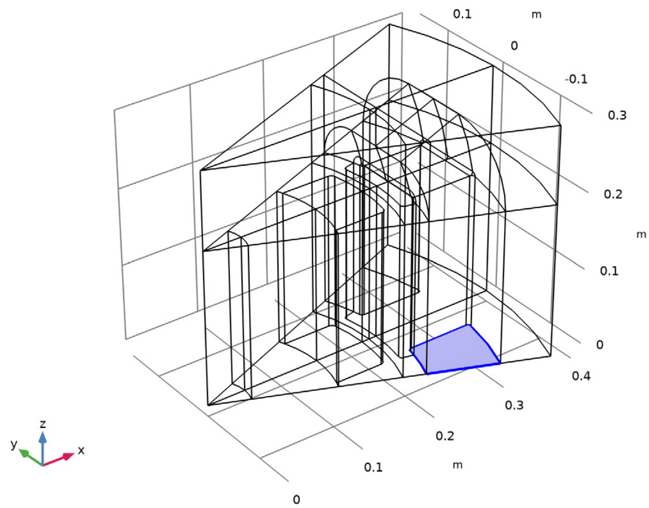
Geometry Analysis 1

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

Output 1

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

2 Select Boundary 76 only (the other boundary at the symmetry cut).



Gauge Fixing for A-field 1

The default solver for the 3D **Rotating Machinery, Magnetic** interface is the direct solver, which gives better performance in time-dependent studies and with the mixed formulation. A direct solver requires that the solution is unique, so it is necessary to apply the **Gauge Fixing** feature. The stationary form of gauge fixing is enforced in order to be able to support regions with zero conductivity.

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

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

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

4 Click **OK**.

Sector Symmetry 1

The **Sector Symmetry** feature sets up the appropriate rotational-periodic coupling at the identity pair.

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 1 (ap1)** 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 `8`.

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

9 Click to expand the **Constraint Settings** section. From the **Constraint** list, choose **Weak constraints**.

Periodic Condition 1

Apply periodic conditions on the periodic boundaries.

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

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

3 From the **Selection** list, choose **Periodic Condition: Rotor**.

4 Locate the **Periodic Condition** section. From the **Type of periodicity** list, choose **Antiperiodicity**.

Periodic Condition 2

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

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

3 From the **Selection** list, choose **Periodic Condition: Stator, Scalar Potential**.

4 Locate the **Periodic Condition** section. From the **Type of periodicity** list, choose **Antiperiodicity**.

Periodic Condition 3

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

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

3 From the **Selection** list, choose **Periodic Condition: Stator, Vector Potential**.

4 Locate the **Periodic Condition** section. From the **Type of periodicity** list, choose **Antiperiodicity**.

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 **Rotating Domains**.

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

5 From the **Rotational velocity expression** list, choose **Constant revolutions per time**.

6 In the *f* text field, type `rpm`.

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**.

Set the electrical conductivity of air and soft iron to a finite small value (1[S/m]) to improve the numerical stability.

MATERIALS

Air (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.

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

3 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

Soft Iron (Without Losses) (mat2)

1 In the **Model Builder** window, click **Soft Iron (Without Losses) (mat2)**.

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

3 In the table, enter the following settings:

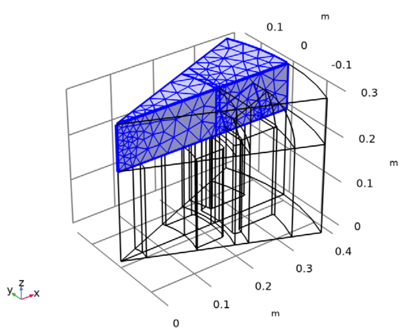
Property	**Variable**	**Value**	**Unit**	**Property group**
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	1[S/m]	S/m	Basic

MESH 1

For curl elements that describe the vector potential **A**, it is important that the mesh is congruent on the two symmetry surfaces, that is, there is a one-to-one mapping between the mesh elements on either side. This eliminates the need for interpolation between mismatching mesh elements which would have a negative impact on accuracy and solver stability. Create a congruent mesh by meshing half of the sector geometry and copying it to the remaining domains. You can also utilize the extrusion along the rotational axis by using a swept mesh, which reduces the size of the problem. Begin by meshing half of the nonextruded domains:

Free Tetrahedral 1

1. In the **Mesh** toolbar, click **Free Tetrahedral**.
2. In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 4, 14, and 21–23 only.
5. Click **Build Selected**.



Swept 1

1. In the **Mesh** toolbar, click **Swept**.
2. In the **Settings** window for **Swept**, locate the **Domain Selection** section.

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

4 Select Domains 2, 6, 8, 10, 13, 16, and 24 only.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

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

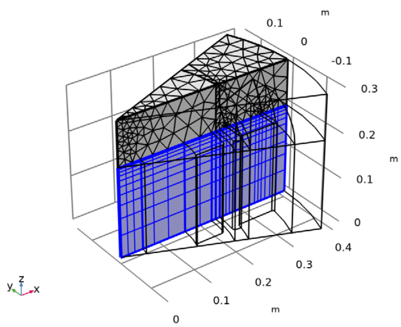
3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type 10.

5 In the **Element ratio** text field, type 10.

6 From the **Growth rate** list, choose **Exponential**.

7 Click **Build Selected**.

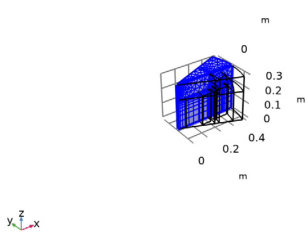


Copy Domain 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Copying Operations> Copy Domain**.

2 Select Domains 2, 4, 6, 8, 10, 13, 14, 16, and 21–24 only.

3 Click the **Go to Default View** button in the **Graphics** toolbar.

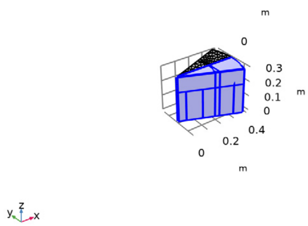


4 In the **Settings** window for **Copy Domain**, locate the **Destination Domains** section.

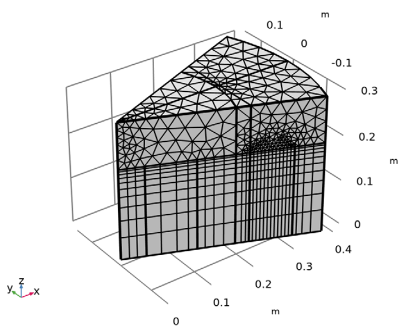
5 Click to select the **Activate Selection** toggle button.

6 Select Domains 1, 3, 5, 7, 9, 11, 12, 15, and 17–20 only.

7 Click the **Go to Default View** button in the **Graphics** toolbar.



8 Right-click **Mesh 1** and choose **Build All**.



STUDY 1

Complete the definition of the study. After the **Coil Geometry Analysis** step added earlier, add a **Stationary** step (to compute the initial values for the transient step) and the main **Time Dependent** step.

Step 2: Stationary

In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

Step 3: 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(0,0.005,0.25)`.

For a time-dependent study with finite conductivity in all domains, it is not necessary to use the gauge fixing. Next, it is disabled to make the model cheaper to solve.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Rotating Machinery, Magnetic (rmm), Controls spatial frame>Gauge Fixing for A-field 1**.

6 Right-click and choose **Disable**.

Solution 1 (sol1)

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

Use a stricter tolerance for the stationary step to compute more accurate initial values. This improves the performance of the time dependent solver. Then operate on the setting of the transient solver so to enforce a tighter convergence at each step.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then 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 **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Dependent Variables 3**.

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

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

8 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>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**.

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

Before proceeding with computing the solution, prepare a plot that is to be shown during solving. Start by adding a selection to the main dataset.

RESULTS

In the **Model Builder** window, expand the **Results** node.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1/ Solution 1 (sol1)**.

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 1, 2, 5, 6, 9, 10, 15, and 16 only.

Reconstruct the complete geometry using **Mirror** and **Sector 3D** datasets.

Sector 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Sector 3D**.
2 In the **Settings** window for **Sector 3D**, locate the **Symmetry** section.
3 In the **Number of sectors** text field, type `8`.
4 Click to expand the **Advanced** section. Select the **Invert phase when rotating** check box.

Complete Geometry, Iron

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.
2 In the **Settings** window for **Mirror 3D**, type `Complete Geometry, Iron` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Sector 3D 1**.
4 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

Magnetic Flux Density (Radial Component)

1 In the **Results** toolbar, click **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Magnetic Flux Density (Radial Component)` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Complete Geometry, Iron**.
4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

   Visualize the solution in the spatial frame (the fixed "laboratory" frame), in which the rotor is moving.

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

6 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

7 Select the **Show units** check box.

Volume 1

1 Right-click **Magnetic Flux Density (Radial Component)** and choose **Volume**.

One way to define the radial component of $\mathbf{B}$, is to write it as $\mathbf{r}\cdot\mathbf{B}$, where $\mathbf{r}$ is a unit vector pointing in the radial direction. You can retrieve it from the cylindrical coordinate system.

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)>Definitions> Cylindrical System 2>Base vector (sys2) r (spatial frame)>sys2.e_r1 - Base vector (sys2) r, x-component**.

Now finish the equation as follows:

3 Locate the **Expression** section. In the **Expression** text field, type `sys2.e_r1*rmm.Bx+ sys2.e_r2*rmm.By`.

4 Select the **Description** check box. In the associated text field, type `Magnetic flux density, r component`.

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Rainbow>Dipole** in the tree.

7 Click **OK**.

8 In the **Settings** window for **Volume**, locate the **Coloring and Style** section.

9 From the **Scale** list, choose **Linear symmetric**.

STUDY 1

Step 3: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 3: 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 **Update at** list, choose **Time steps taken by solver**.

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

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

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

Now that the plot has been prepared and set to be shown during solving, proceed by computing the solution.

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

RESULTS

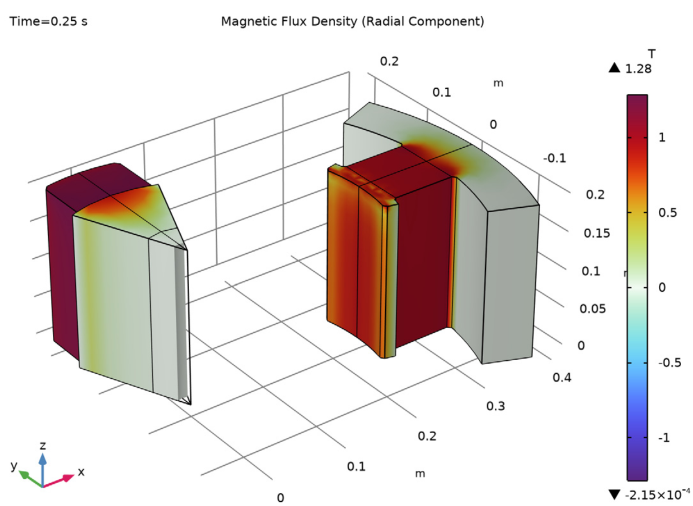
Magnetic Flux Density (Radial Component)

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

2 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

3 In the **Magnetic Flux Density (Radial Component)** toolbar, click **Plot**.

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



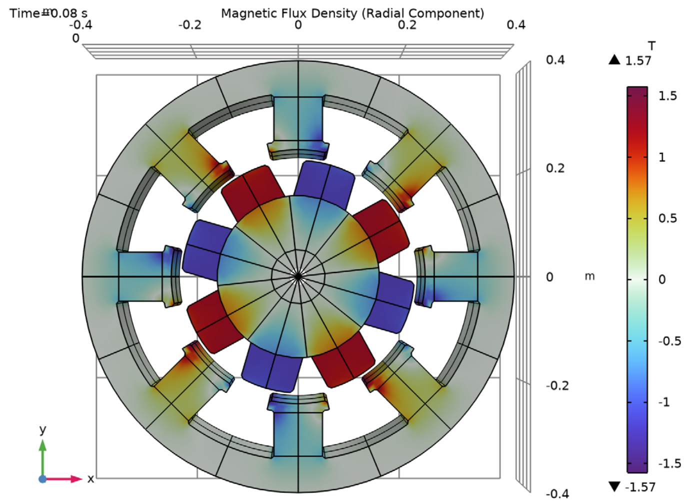
5 From the **Dataset** list, choose **Complete Geometry, Iron**.

6 From the **Time (s)** list, choose **0.08**.

7 In the **Magnetic Flux Density (Radial Component)** toolbar, click **Plot**.

8 Click the **Go to XY View** button in the **Graphics** toolbar.

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



Induced Coil Voltage

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

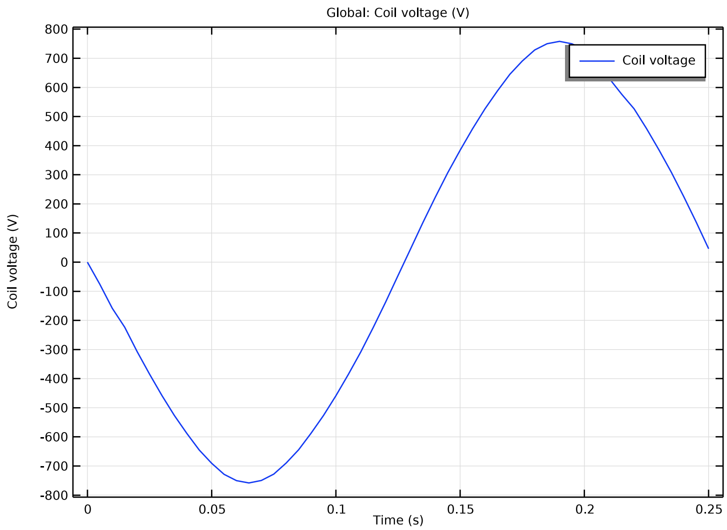
2 In the **Settings** window for **1D Plot Group**, type `Induced Coil Voltage` in the **Label** text field.

Global 1

1 Right-click **Induced Coil Voltage** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Rotating Machinery, Magnetic (Magnetic Fields)>Coil parameters>rmm.VCoil_1 - Coil voltage - V**.

3 In the **Induced Coil Voltage** toolbar, click **Plot**.



Next, add the **Time to Frequency Losses** study to compute the loss.

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 2

Step 1: Time to Frequency Losses

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

2 From the **Input study** list, choose **Study 1, Time Dependent**.

3 In the **Electrical period** text field, type `0.25`.

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

RESULTS

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 **Study 2/Solution 4 (sol4)**.

4 Select Domains 1, 2, 5, 6, 9, 10, 15, and 16 only.

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

Expression	Unit	Description
rmm.Qh*16	W	Loss power

6 Click **Evaluate**.

The computed loss power is close to the value computed from the 2D model.

Animation 1

Finally, visualize the rotation of the generator by animating the solution.

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

2 In the **Settings** window for **Animation**, locate the **Playing** section.

3 From the **Repeat** list, choose **Number of iterations**.

4 In the **Number of iterations** text field, type 10.

5 Click the **Go to Default View** button in the **Graphics** toolbar.

6 In the **Graphics** toolbar, use the **Play** and **Stop** buttons to control the animation.