



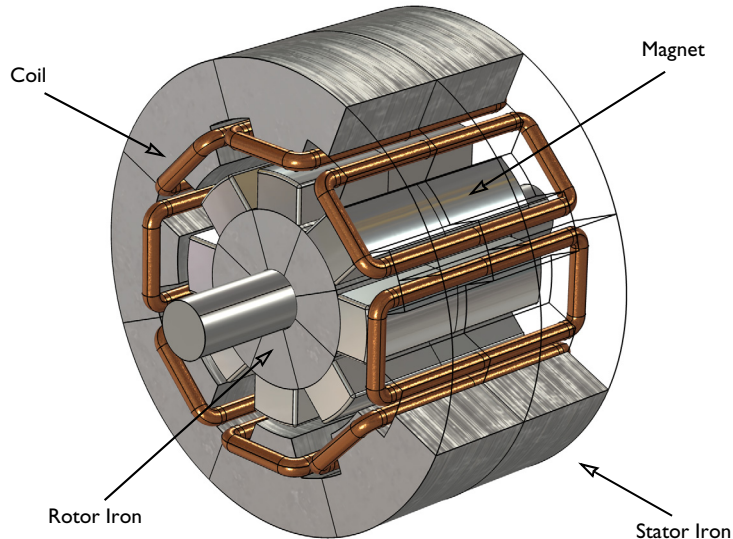
# Generator in 2D

## *Introduction*

---

This example shows how the circular motion of a rotor with permanent magnets generates an induced EMF in a stator winding. The generated voltage is calculated as a function of time during the rotation. The model also shows the influence on the voltage from material parameters, rotation velocity, and number of turns in the winding.

The center of the rotor consists of annealed medium carbon steel, which is a material with a high relative permeability. The center is surrounded with several blocks of a permanent magnet made of sintered NdFeB, creating a strong magnetic field. The stator is made of the same permeable material as the center of the rotor, confining the field in closed loops through the winding. The winding is wound around the stator poles. [Figure 1](#) shows the generator with part of the stator sliced in order to show the winding and the rotor.



*Figure 1: Drawing of a generator showing how the rotor, stator, and stator winding are constructed. The winding is also connected between the loops, interacting to give the highest possible voltage.*

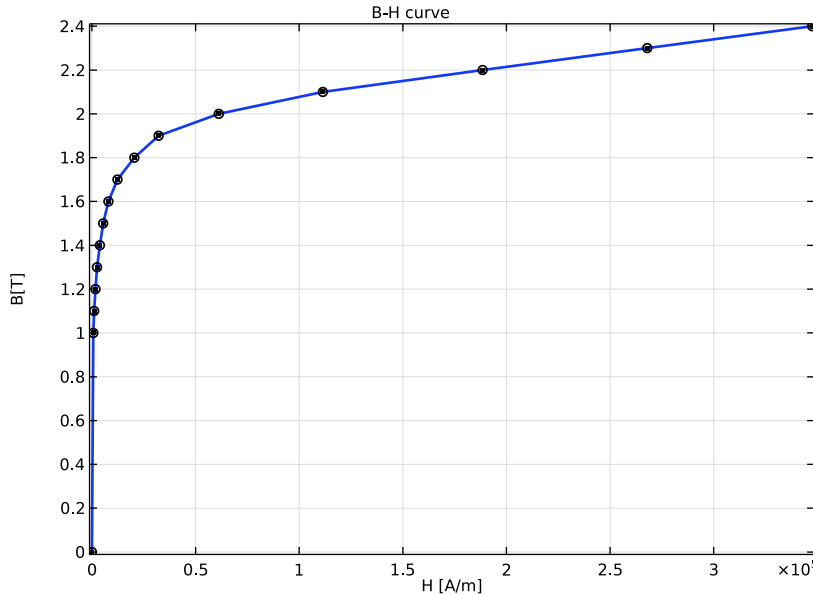
The COMSOL Multiphysics model of the generator is a time-dependent 2D problem on a cross section through the generator. This is a true time-dependent model where the motion of the magnetic sources in the rotor is accounted for in the boundary condition between the stator and rotor geometries. Thus, there is no Lorentz term in the equation, resulting in the PDE

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

where the magnetic vector potential only has a  $z$  component.

Rotation is modeled using a ready-made physics interface for rotating machinery. The center part of the geometry, containing the rotor and part of the air-gap, rotates relative to the coordinate system of the stator. The rotor and the stator are imported as two separate geometry objects, so it is possible to use an assembly (see *Finalizing the Geometry* in the *COMSOL Multiphysics Reference Manual* for details). This has several advantages: the coupling between the rotor and the stator is done automatically, the parts are meshed independently, and it allows for a discontinuity in the vector potential at the interface between the two geometry objects (called slits). The rotor problem is solved in a rotating coordinate system where the rotor is fixed (the rotor frame), whereas the stator problem is solved in a coordinate system that is fixed with respect to the stator (the stator frame). An identity pair connecting the rotating rotor frame with the fixed stator frame is created between the rotor and the stator. The identity pair enforces continuity for the vector potential in the global fixed coordinate system (the stator frame).

The stator and center of the rotor are made of annealed medium-carbon steel (soft iron), which is implemented in COMSOL Multiphysics as an interpolation function of the B-H curve of the material; see [Figure 2](#).



*Figure 2: The norm of the magnetic flux,  $|\mathbf{B}|$ , versus the norm of the magnetic field,  $|\mathbf{H}|$ , for the rotor and stator materials.*

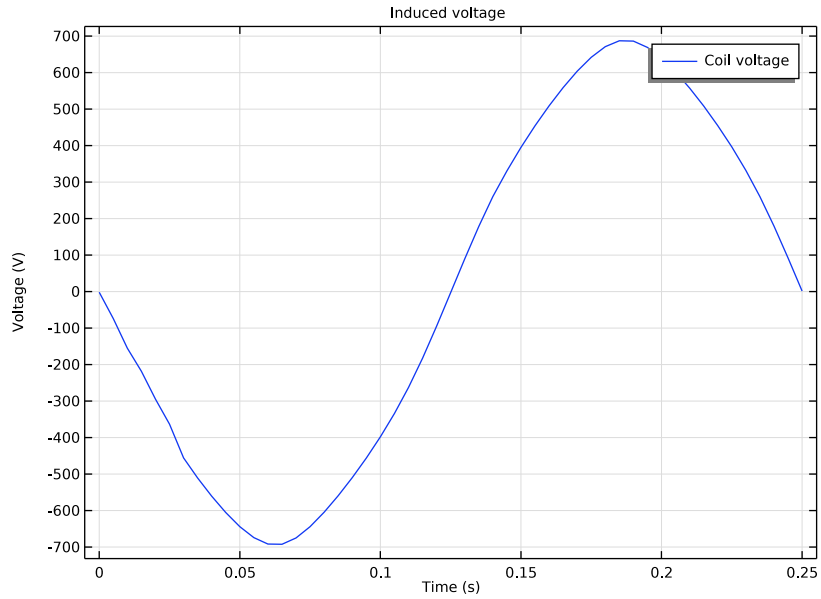
The generated voltage is computed automatically with the use of the Coil feature. By selecting all the coils and putting them in a Coil group, it takes into account the topology of the windings that are arranged in series. In this way, the output voltage is the sum of voltages generated by each coils even though they are not geometrically connected in the 2D model.

The resistive and iron losses are computed with the use of a Loss Calculation subfeature and a Time to Frequency Losses study. In this 2D model, the magnets are modeled by using the Coil feature, which allows for the calculation of circulating currents and resistive losses. The settings impose a zero net perpendicular current in each of the magnet by activating the Coil group checkbox. For the iron part of the generator, the Bertotti loss model is selected to compute the cycle averaged loss power density. A surface integration is then made to compute the total loss power of the generator. The generated voltage and the losses are compared with the 3D counterpart of this model, i.e., *Modeling of an Electric Generator in 3D*.

## Results and Discussion

---

The generated voltage in the rotor winding is close to a sinusoidal signal. At a rotation speed of 60 rpm the voltage has an amplitude slightly lower 700 V in total; see [Figure 3](#).



*Figure 3: The generated voltage over one quarter of a revolution. This simulation used a 100-turn winding.*

The norm of the magnetic flux,  $|\mathbf{B}|$ , and the  $\mathbf{B}$  field lines are shown below in Figure 4 at time 0.20 s.

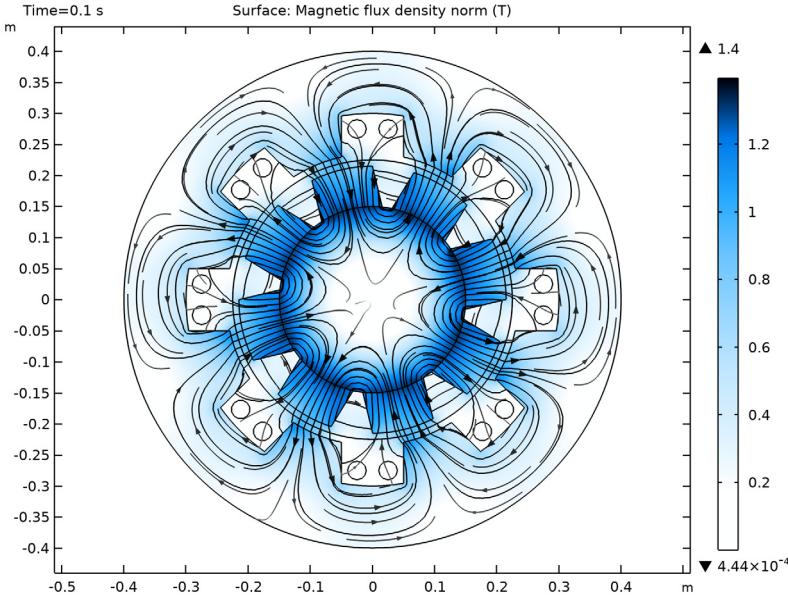
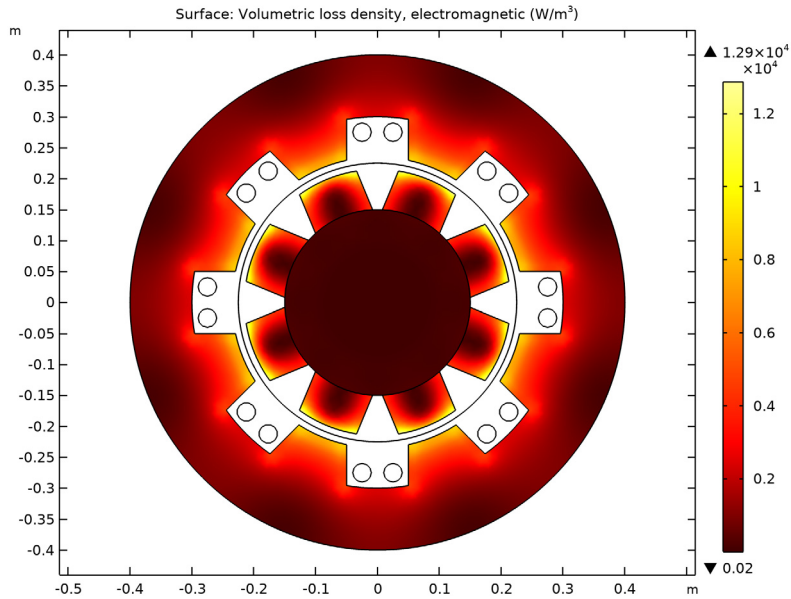


Figure 4: The norm and the field lines of the magnetic flux after 0.2 s of rotation. Note the darker regions, which indicate the position of the permanent magnets in the rotor.

The cycle averaged losses power density is shown below in [Figure 5](#).



*Figure 5: The volumetric loss density of the generator.*

---

**Application Library path:** ACDC\_Module/Motors\_and\_Actuators/generator\_2d


---

### *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**.

## GLOBAL DEFINITIONS

### *Parameters I*


- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
L	0.4[m]	0.4 m	Length of generator
rpm	60[1/min]	1 l/s	Rotational speed of rotor
d_wire	3[mm]	0.003 m	Diameter of wire in the winding
N	100	100	Number of turns in the winding

## GEOMETRY I

### *Import I (impl)*

Import the geometry of the generator cross section from an external CAD file.


- 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 generator\_2d.mphbin.
- 5 Click **Import**.

### *Form Union (fin)*

The geometry you imported is composed by two objects, an inner part (corresponding to the rotor) and an outer part (the stator). Create an assembly from these objects.

- 1 In the **Model Builder** window, under **Component I (comp1)>Geometry I** 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**.

In this way, an identity pair connecting the shared boundaries has been automatically created.



## DEFINITIONS

*Cylindrical System 2 (sys2)*

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

The cylindrical coordinate system you just added will be used to define the field of the permanent magnets.

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

## MATERIALS

*Soft Iron (Without Losses) (mat2)*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Soft Iron (Without Losses) (mat2)**.
- 2 Select Domains 2 and 28 only.

*N50 (Sintered NdFeB) (mat3)*

- 1 In the **Model Builder** window, click **N50 (Sintered NdFeB) (mat3)**.
- 2 Select Domains 20–27 only.

## 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.

Next, use a Rotating Domain feature to specify the rotor's rotational velocity.

## DEFINITIONS


### *Rotating Domain 1*

- 1 In the **Definitions** toolbar, click  **Moving Mesh** and choose **Rotating Domain**.
- 2 Select Domains 19–28 only.
- 3 In the **Settings** window for **Rotating Domain**, locate the **Rotation** section.
- 4 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)

The permanent magnets on the rotor have a radial field. Use the cylindrical coordinate system specified earlier to define it.

### *Permanent Magnets Outward*

- 1 In the **Physics** toolbar, click  **Domains** and choose **Coil**.
- 2 In the **Settings** window for **Coil**, type Permanent Magnets Outward in the **Label** text field.
- 3 Select Domains 20, 23, 24, and 27 only.
- 4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.
- 5 Locate the **Coil** section. Select the **Coil group** check box.
- 6 In the  $I_{\text{coil}}$  text field, type 0[A].
- 7 Locate the **Constitutive Relation B-H** section. From the **Magnetization model** list, choose **Remanent flux density**.

### *Loss Calculation 1*

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


### *Permanent Magnets Inward*

- 1 In the **Model Builder** window, right-click **Permanent Magnets Outward** and choose **Duplicate**.

- 2 In the **Settings** window for **Coil**, type Permanent Magnets Inward in the **Label** text field.
- 3 Select Domains 21, 22, 25, and 26 only.
- 4 Locate the **Constitutive Relation B-H** section. Specify the **e** vector as


-1	r
0	phi
0	a

#### *Iron*


- 1 In the **Physics** toolbar, click  **Domains** and choose **Ampère's Law**.
- 2 In the **Settings** window for **Ampère's Law**, locate the **Constitutive Relation B-H** section.
- 3 From the **Magnetization model** list, choose **B-H curve**.
- 4 Select Domains 2 and 28 only.
- 5 In the **Label** text field, type Iron.

Add the **Loss Calculation** subfeature to compute iron losses.

#### *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**.

#### *Coil 3*

- 1 In the **Physics** toolbar, click  **Domains** and choose **Coil**.
- 2 Select Domains 3–18 only.
- 3 In the **Settings** window for **Coil**, locate the **Coil** section.
- 4 From the **Conductor model** list, choose **Homogenized multiturn**.
- 5 In the  $I_{\text{coil}}$  text field, type 0[A].
- 6 Locate the **Homogenized Multiturn Conductor** section. In the  $N$  text field, type N.
- 7 From the **Coil wire cross-section area** list, choose **From round wire diameter**.
- 8 In the  $d_{\text{coil}}$  text field, type d\_wire.
- 9 Locate the **Coil** section. Select the **Coil group** check box.


#### *Reversed Current Direction 1*

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Reversed Current Direction**.

2 Select Domains 3, 4, 9–12, 17, and 18 only.


The problem will be solved separately in the fixed frame of the stator and the rotating frame of the rotor. Apply a Continuity feature on the shared boundaries (that are connected by **Identity Pair I**).

#### *Continuity I*

- 1 In the **Physics** toolbar, click  **Pairs** and choose **Continuity**.
- 2 In the **Settings** window for **Continuity**, locate the **Pair Selection** section.
- 3 Under **Pairs**, click **+ Add**.
- 4 In the **Add** dialog box, select **Identity Boundary Pair I (apI)** in the **Pairs** list.
- 5 Click **OK**.

#### **STUDY I**

#### *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).

The problem is set and it would be essentially ready to solve. In the next section some adjustments to physics, mesh, and solver settings are presented. These offer a more stable solution which is also found in a shorter time.

#### *Suggestion to make solution faster and more stable*

---


There are several settings which, when applied to model like this, make solution faster and more stable. Even though not all of these are required for the presented simple generator model, all of these may turn to be useful for more complex problems. In particular, these settings are being applied:

- use of **Linear** elements for **Discretization** as this is offering a better alternative in models with magnetic saturation;
- use **Weak constraints** in the **Continuity** condition applied where the pair with sliding meshes is active; this offers an higher precision for a given mesh;
- use, in the **Continuity** identity pair, a destination mesh finer than the source as this simplifies continuity constraint handling;

- use tighter tolerances than default in stationary preliminary step as this will simplify the starting of the subsequent transient which is in general more demanding;
- use **Initial value basedScaling** of physical variables in time dependent study step; eventually set also scaling of rotation angle variable; such settings will make the time dependent solver able to better choose the time step;
- set **Update Jacobian** to **On every iteration** as this makes each of the time dependent solver step more stable;
- set **Exclude algebraic** in error estimate as this will enable time stepper to better balance equation among the actively inductive contributions and those corresponding to instantaneously adapting fields;
- set **Steps taken by solver** to **Intermediate** or **Strict** as this will offer a better value also for quantities which contains time derivatives as the electric potentials on the **Coil**.

## DEFINITIONS

### *Identity Boundary Pair 1 (ap1)*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Identity Boundary Pair 1 (ap1)**.
- 2 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.
- 3 Click  **Create Selection**.
- 4 In the **Create Selection** dialog box, type destination in the **Selection name** text field.
- 5 Click **OK**.

## MESH 1


### *Free Triangular 1*

In the **Mesh** toolbar, click  **Free Triangular**.


### *Size*

- 1 In the **Model Builder** window, click **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Coarser**.
- 4 Click the **Custom** button.
- 5 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 0.015.

### *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 **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 From the **Selection** list, choose **destination**.
- 5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.
- 6 Click  **Build All**.

### **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**.
- 5 Click the  **Show More Options** button in the **Model Builder** toolbar.
- 6 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.
- 7 Click **OK**.


### *Continuity 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Rotating Machinery, Magnetic (rmm)** click **Continuity 1**.
- 2 In the **Settings** window for **Continuity**, click to expand the **Constraint Settings** section.
- 3 Select the **Use weak constraints** check box.

### **STUDY 1**

#### *Solution 1 (sol1)*

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Stationary Solver 1**.
- 3 In the **Settings** window for **Stationary Solver**, locate the **General** section.
- 4 In the **Relative tolerance** text field, type  $1e-5$ .
- 5 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Dependent Variables 2**.

- 6 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.
- 7 From the **Method** list, choose **Initial value based**.
- 8 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Time-Dependent Solver 1**.
- 9 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- 10 From the **Steps taken by solver** list, choose **Strict**.
- 11 Find the **Algebraic variable settings** subsection. From the **Error estimation** list, choose **Exclude algebraic**.
- 12 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** click **Fully Coupled 1**.
- 13 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
- 14 From the **Jacobian update** list, choose **On every iteration**.
- 15 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** click **Direct**.
- 16 In the **Settings** window for **Direct**, locate the **General** section.
- 17 From the **Solver** list, choose **PARDISO**.
- 18 In the **Study** toolbar, click  **Compute**.

### *Postprocessing the results*

---

Now, plot the solution in the spatial frame (the stator's fixed frame) at time  $t = 0.2$  s.

- 1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Magnetic Flux Density Norm (rmm)**.
- 2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **0.1**.


#### *Contour 1*

- 1 In the **Model Builder** window, expand the **Magnetic Flux Density Norm (rmm)** node, then click **Contour 1**.
- 2 In the **Settings** window for **Contour**, locate the **Levels** section.
- 3 In the **Total levels** text field, type 12.


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

As shown in [Figure 4](#), it shows the rotor position at  $t = 0.2$  s, the magnetic flux density norm and magnetic flux lines. Next, plot the induced EMF in a quarter of the cycle.

#### *Induced Coil Voltage*


- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Induced Coil Voltage 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 Induced voltage.
- 5 Locate the **Plot Settings** section. Select the **x-axis label** check box.
- 6 In the associated text field, type Time (s).
- 7 Select the **y-axis label** check box.
- 8 In the associated text field, type Voltage (V).

#### *Global I*

- 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 I (comp I)> Rotating Machinery, Magnetic (Magnetic Fields)>Coil parameters>rmm.VCoil\_3 - Coil voltage - V**.
- 3 In the **Induced Coil Voltage** toolbar, click  **Plot**.  
The resulting plot is [Figure 3](#).

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

#### **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

*Cycle Averaged Losses (rmm)*

The resulting plot is [Figure 5](#).

Finally, compute the total iron loss power.

*Surface Integration I*

- 1 In the **Results** toolbar, click  **More Derived Values** and choose **Integration>Surface Integration**.
- 2 Select Domains 2 and 20–28 only.
- 3 In the **Settings** window for **Surface Integration**, locate the **Data** section.
- 4 From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.
- 5 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Rotating Machinery, Magnetic (Magnetic Fields, No Currents)>Heating and losses>rmm.Qh - Volumetric loss density, electromagnetic - W/m<sup>3</sup>**.
- 6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
rmm.Qh*L	W	Total loss power

- 7 Click  **Evaluate**.

The computed total loss power is about 303 W.

