Created in COMSOL Multiphysics 6.2

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.

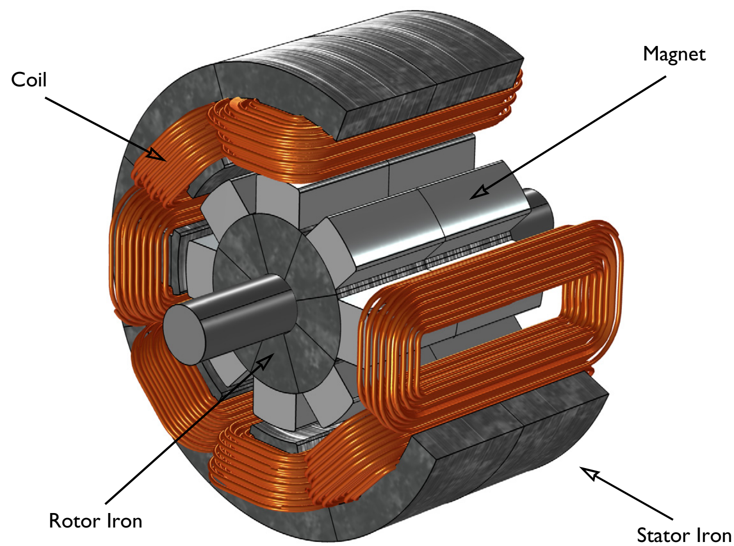


Figure 1: Drawing of a generator showing how the rotor, stator, and stator winding are constructed. The coils are connected in series in order to give the highest possible voltage.

Modeling in COMSOL Multiphysics

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 built 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.

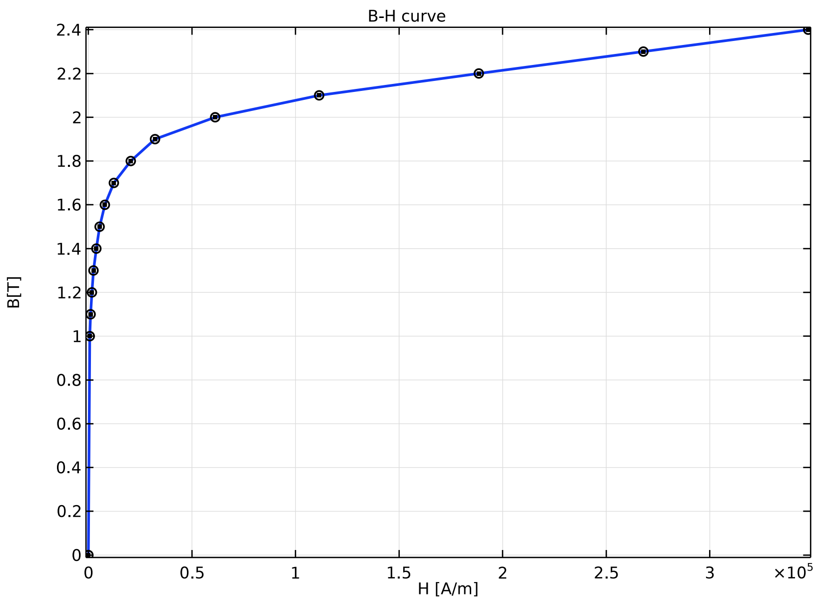


Figure 2: The norm of the magnetic flux, $|\mathbf{B}|$, versus the norm of the magnetic field, $|\mathbf{H}|$, for the rotor and stator materials.

The generated voltage is computed automatically with the use of the Coil feature. In this simplified example all coils are connected in series, making out a single stator winding. The series connection of coils is realized in COMSOL Multiphysics by adding coil domains to a Coil group. In this way, the output voltage is the sum of voltages generated by each coil 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 induced currents and resistive losses. The settings impose a zero net perpendicular current in each of the magnets by activating the Coil group check box. 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, that is, 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 of 700 V in total; see Figure 3.

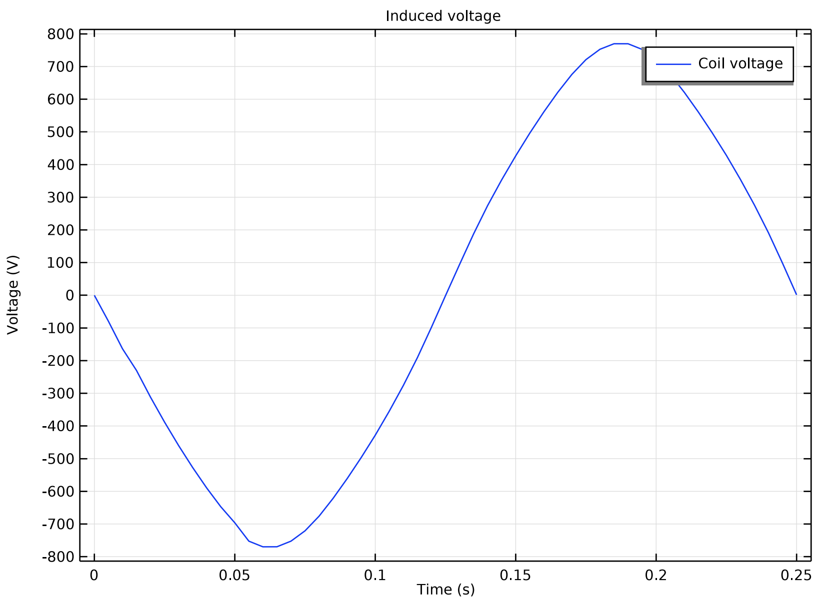


Figure 3: The generated voltage over one quarter of a revolution. This simulation used a 100-turn winding.

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

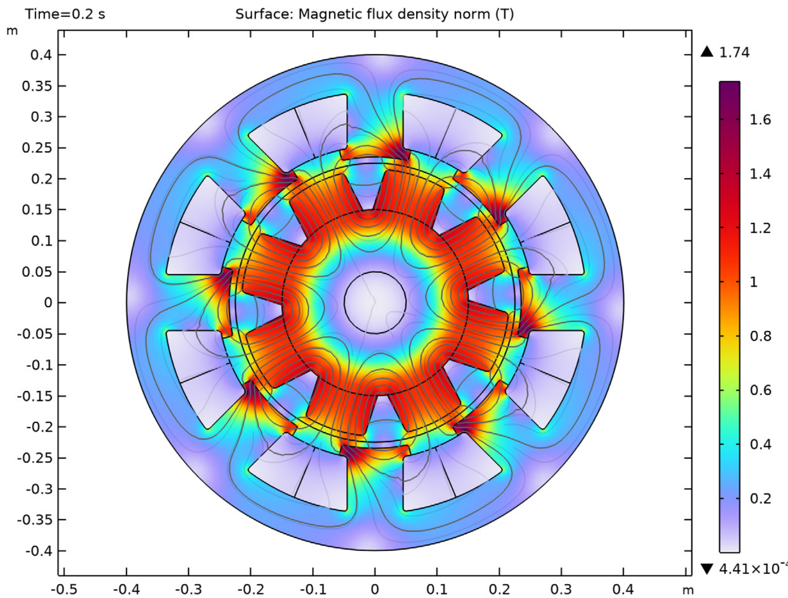


Figure 4: The norm and the field lines of the magnetic flux density after 0.2 s of rotation. Note the darker regions, which indicate the position of the permanent magnets in the rotor.

The cycle averaged losses power density is shown below in Figure 5.

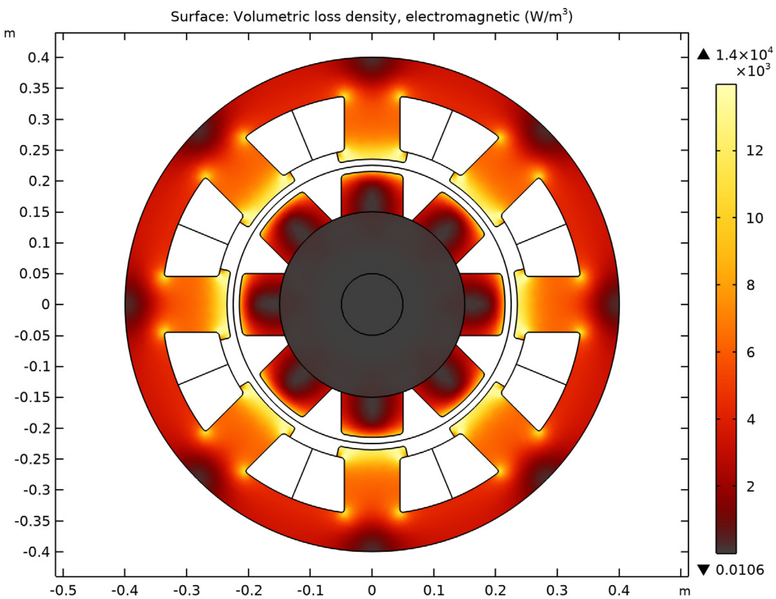


Figure 5: The volumetric loss density of the generator.

Application Library path: `ACDC_Module/Devices,_Motors_and_Generators/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 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
L	0.4[m]	0.4 m	Length of generator
rpm	60[1/min]	1 1/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

Next, build the generator using rotor and stator parts from the geometry part library. Initialize the parts, and tick the selections that are predefined to make it convenient to assign material properties and magnetization direction.

PART LIBRARIES

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

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

3 Click **Add to Geometry**.

PART LIBRARIES

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

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

3 In the **Part Libraries** window, select **AC/DC Module>Rotating Machinery 2D>Stators>External>slotted_external_stator_2d** in the tree.

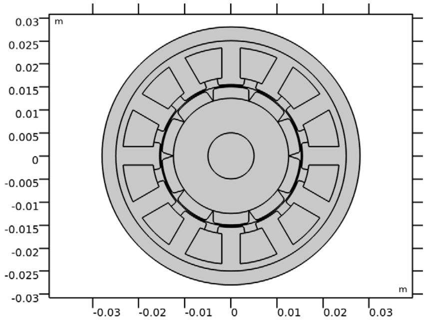
4 Click **Add to Geometry**.

GEOMETRY 1

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

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

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



Internal Rotor – Surface Mounted Magnets 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Internal Rotor – Surface Mounted Magnets 1 (pi1)**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
number_of_poles	8	8	Number of magnetic poles in rotor
number_of_modeled_poles	8	8	Number of magnetic poles included in the geometry
shaft_diam	`10[cm]`	0.1 m	Diameter of the shaft
rotor_diam	`30[cm]+2*6.5[cm]`	0.43 m	Diameter of the rotor
cont_diam	`0.45[m]`	0.45 m	Diameter of the stator-rotor continuity interface
magnet_h	`6.5[cm]`	0.065 m	Height of the magnets
magnet_w	`10 [cm]`	0.1 m	Width of the magnets (set to 0 to use all available space)
magnet_fillet_size	`0.6[cm]`	0.006 m	Radius of magnet fillet

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

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

External Stator – Slotted 1 (pi2)

1 In the **Model Builder** window, click **External Stator – Slotted 1 (pi2)**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
number_of_slots	8	8	Number of slots in stator
number_of_modeled_slots	8	8	Number of slots included in the geometry
backiron_th	`6 [cm]`	0.06 m	Thickness of back-iron
stator_diam	`80 [cm]`	0.8 m	Diameter of the stator
external_air_size	`0[cm]`	0 m	Size of air external to stator
cont_diam	`45[cm]`	0.45 m	Diameter of the stator-rotor continuity interface
shoe_h	`1.75[cm]`	0.0175 m	Height of the shoe
shoe_w	`11.5[cm]`	0.115 m	Width of the shoe
tooth_h	`10.5[cm]`	0.105 m	Height of the tooth
tooth_w	`9[cm]`	0.09 m	Width of the tooth
shoe_fillet_size	`0.4[cm]`	0.004 m	Radius of the shoe fillet
slot_outer_fillet_size	`0.4[cm]`	0.004 m	Radius of the outer slot fillet

Name	Expression	Value	Description
slot_inner_fillet_size	`0.4[cm]`	0.004 m	Radius of the inner slot fillet
slot_winding_type	2	2	Slot winding type: 1-No partition, 2-Radial partition, 3-Azimuthal partition, 4-Radial and azimuthal partition.

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

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

DEFINITIONS

Iron

1. In the **Definitions** toolbar, click **Union**.
2. In the **Settings** window for **Union**, locate the **Input Entities** section.
3. Under **Selections to add**, click **Add**.
4. In the **Add** dialog box, in the **Selections to add** list, choose **Shaft (Internal Rotor – Surface Mounted Magnets 1)**, **Rotor iron (Internal Rotor – Surface Mounted Magnets 1)**, and **Stator iron (External Stator – Slotted 1)**.
5. Click **OK**.
6. In the **Settings** window for **Union**, type `Iron` in the **Label** text field.

Iron and Magnets

1. In the **Definitions** toolbar, click **Union**.
2. In the **Settings** window for **Union**, locate the **Input Entities** section.
3. Under **Selections to add**, click **Add**.
4. In the **Add** dialog box, in the **Selections to add** list, choose **Iron** and **Rotor magnets (Internal Rotor – Surface Mounted Magnets 1)**.
5. Click **OK**.
6. In the **Settings** window for **Union**, type `Iron and Magnets` in the **Label** text field.

Create an assembly from the two geometry objects, connected by a pair boundary.

GEOMETRY 1

Form Union (fin)

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

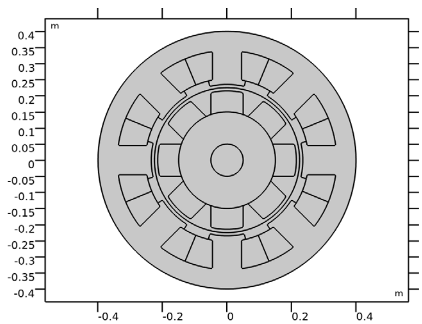
2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

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

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

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

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



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 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3 From the **Selection** list, choose **Iron**.

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 **Rotor magnets (Internal Rotor – Surface Mounted Magnets 1)**.

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.

COMPONENT 1 (COMP1)

Rotating Domain 1

1 In the **Physics** toolbar, click **Moving Mesh** and choose **Rotating Domain**.
2 In the **Settings** window for **Rotating Domain**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **All (Internal Rotor – Surface Mounted Magnets 1)**.
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`.

To define the remanent flux direction of all magnets, use the Conducting Magnet feature.

ROTATING MACHINERY, MAGNETIC (RMM)

Conducting Magnet 1

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

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

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

4 Locate the **Magnet** section. From the **Pattern type** list, choose **Circular pattern**.

5 From the **Type of periodicity** list, choose **Alternating**.

North 1

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

2 Select Boundary 220 only.

South 1

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

2 Select Boundary 226 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**.

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

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 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 slots (External Stator – Slotted 1)**.

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

5 In the I_{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_{wire} 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 1, 4, 8, 10, 11, 13, 16, and 18 only.

STUDY 1

Step 2: 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.

Suggestions for How to Make Solving Faster and More Stable

There are several settings that, when applied to models like this one, make solving faster and more stable. Even though not all of the settings are required for this simple generator model, they can be useful for more complex problems. More specifically, these settings are applied:

- Use **Linear** elements for **Discretization** as this offers 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 a 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 the default in the stationary preliminary step as this will simplify the starting of the subsequent transient, which is in general more demanding

- Use **Initial value based Scaling** of physical variables in the time-dependent study step; also consider setting the scaling of rotation angle variable to make the time-dependent solver better able to choose the time step
- Set **Nonlinear method** to **Automatic (Newton)** as this makes each of the time-dependent solver steps more stable
- Set **Exclude algebraic** in the error estimate as this will enable the time stepper to better balance equations 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 that contain time derivatives, like 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 From the **Constraint** list, choose **Weak constraints**.

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 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** click **Fully Coupled 1**.

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

12 From the **Nonlinear method** list, choose **Automatic (Newton)**.

13 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** click **Direct**.

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

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

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

Postprocessing the Results

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

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

2 From the **Time (s)** list, choose **0.2**.

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

Figure 4 shows the rotor position at $t = 0.2$ s, as well as the magnetic flux density norm and magnetic flux density 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 **1D Plot Group**.

2 In the **Settings** window for **1D 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.

6 Select the **x-axis label** check box. In the associated text field, type `Time (s)`.

7 Select the **y-axis label** check box. In the associated text field, type `Voltage (V)`.

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**.

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.

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

Cycle Averaged Losses (rmm)

The resulting plot is Figure 5.

Finally, compute the total iron loss power.

Surface Integration 1

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Integration> Surface Integration**.

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

3 From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.

4 Locate the **Selection** section. From the **Selection** list, choose **Iron and Magnets**.

5 Click **Replace 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/m³**.

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 455 W.