Created in COMSOL Multiphysics 5.3

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 samarium and cobalt, 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.

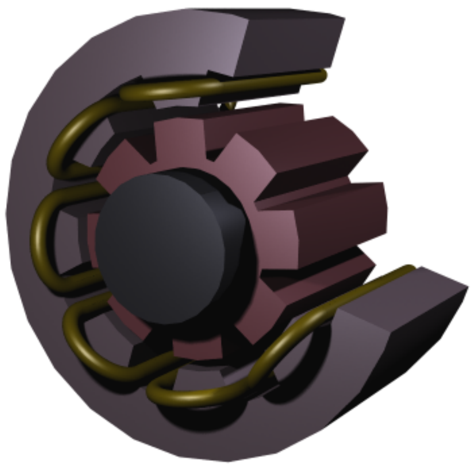


Figure 1: Drawing of a generator showing how the rotor, stator, and stator winding are constructed. The winding is also connected between the loops, interacting to give the highest possible voltage.

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 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. The function can be used in the domain settings. Usually B-H curves are specified as $|\mathbf{B}|$ versus $|\mathbf{H}|$, but the rotating machinery, magnetic interface must have $|\mathbf{H}|$ versus $|\mathbf{B}|$. It is therefore important that the H-data is entered as $f(x)$-data of the interpolation function and the B-data entered as x-data. This relationship for |**H**| is predefined for the material Soft Iron (without losses) in the materials library that is shipped with the AC/DC Module.

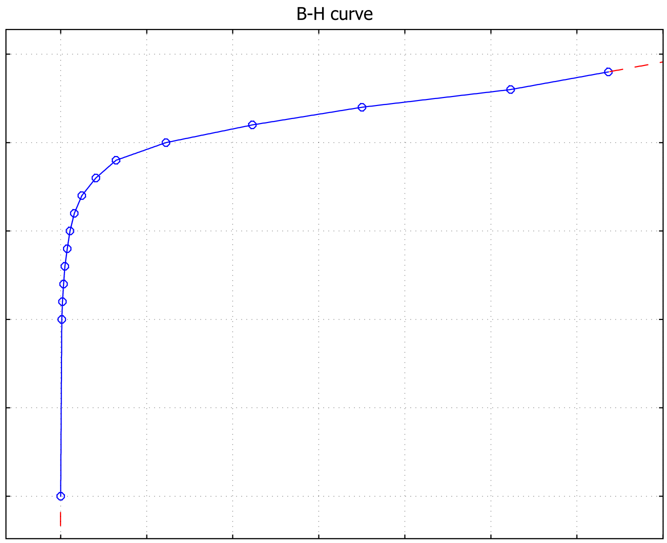


Figure 2: The norm of the magnetic flux, |**B**|, *versus the norm of the magnetic field,* |**H**|, *for the rotor and stator materials. The inverse of this curve is used in the calculation.*

The generated voltage is computed as the line integral of the electric field, **E**, along the winding. Because the winding sections are not connected in the 2D geometry, a proper line integral cannot be carried out. A simple approximation is to neglect the voltage contributions from the ends of the rotor, where the winding sections connect. The voltage is then obtained by taking the average z component of the **E** field for each winding cross-section, multiplying it by the axial length of the rotor, and taking the sum over all winding cross sections.

$$V_i = \mathrm{NN} \sum_{\text{windings}} \frac{L}{A} \int E_z dA$$

where L is the length of the generator in the third dimension, NN is the number of turns in the winding, and A is the total area of the winding cross-section.

Results and Discussion

The generated voltage in the rotor winding is a sinusoidal signal. At a rotation speed of 60 rpm the voltage has an amplitude slightly above 4 V for a single turn winding; see Figure 3.

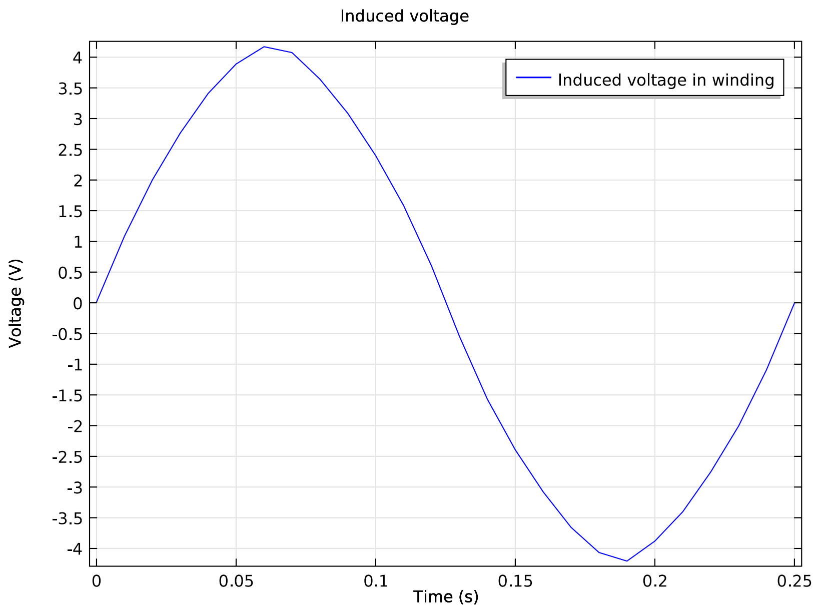


Figure 3: The generated voltage over one quarter of a revolution. This simulation used a single-turn winding.

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

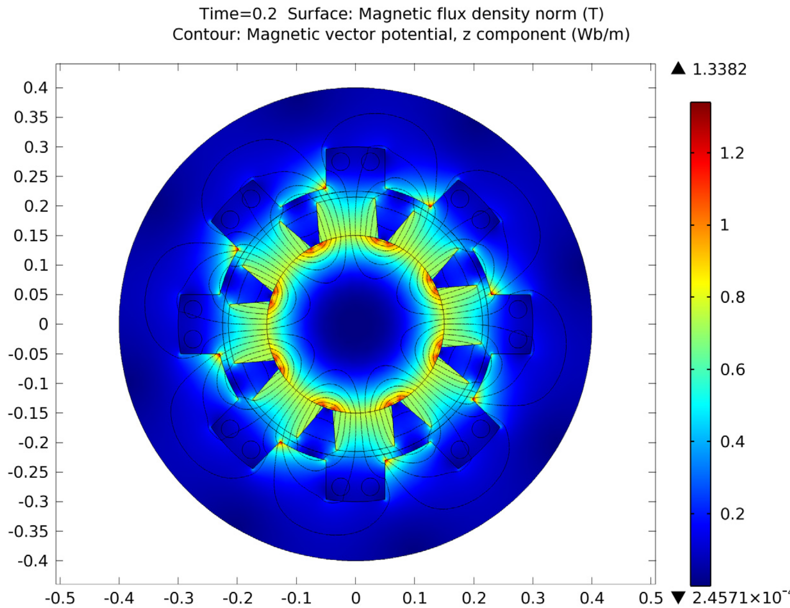


Figure 4: The norm and the field lines of the magnetic flux after 0.2 s of rotation. Note the brighter regions, which indicate the position of the permanent magnets in the rotor.

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>Rotating Machinery, Magnetic (rmm)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1. On the **Home** toolbar, click **Parameters**.
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`
A	`pi*(3[cm]/2)^2`	7.069E-4 m²	`Area of single circular conductor coil`

GEOMETRY 1

Import 1 (imp1)

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

1. On 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 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 On 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 On 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 On 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.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **AC/DC>Soft Iron (without losses)**.

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

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.

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

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`.

Apply a Prescribed Rotational Velocity feature to the rotor, and specify its rotational velocity using the parameter `rpm`.

Prescribed Rotational Velocity 1

1 Right-click **Component 1 (comp1)>Rotating Machinery, Magnetic (rmm)** and choose **Prescribed Rotational Velocity**.

2 Select Domains 19–28 only.

3 In the **Settings** window for **Prescribed Rotational Velocity**, locate the **Prescribed Rotational Velocity** section.

4 In the rps text field, type `rpm`.

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

Ampère's Law 2

1 In the **Model Builder** window, right-click **Rotating Machinery, Magnetic (rmm)** and choose the domain setting **Vector Potential>Ampère's Law**.

2 Select Domains 20, 23, 24, and 27 only.

3 In the **Settings** window for **Ampère's Law**, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

5 Locate the **Magnetic Field** section. From the **Constitutive relation** list, choose **Remanent flux density**.

6 Specify the $\mathbf{B}_r$ vector as

0.84[T]	r
0	phi
0	a

7 Right-click **Component 1 (comp1)>Rotating Machinery, Magnetic (rmm)>Ampère's Law 2** and choose **Rename**.

8 In the **Rename Ampère's Law** dialog box, type `Permanent Magnets Outward` in the **New label** text field.

9 Click **OK**.

The other four magnets are reversed.

Ampère's Law 3

1 In the **Model Builder** window, right-click **Rotating Machinery, Magnetic (rmm)** and choose the domain setting **Vector Potential>Ampère's Law**.

2 Select Domains 21, 22, 25, and 26 only.

3 In the **Settings** window for **Ampère's Law**, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

5 Locate the **Magnetic Field** section. From the **Constitutive relation** list, choose **Remanent flux density**.

6 Specify the $\mathbf{B}_\mathbf{r}$ vector as

-0.84[T]	r
0	phi
0	a

7 Right-click **Component 1 (comp1)>Rotating Machinery, Magnetic (rmm)>Ampère's Law 3** and choose **Rename**.

8 In the **Rename Ampère's Law** dialog box, type `Permanent Magnets Inward` in the **New label** text field.

9 Click **OK**.

Ampère's Law 4

1 In the **Model Builder** window, right-click **Rotating Machinery, Magnetic (rmm)** and choose the domain setting **Vector Potential>Ampère's Law**.

2 In the **Settings** window for **Ampère's Law**, locate the **Magnetic Field** section.

3 From the **Constitutive relation** list, choose **HB curve**.

4 Select Domains 2 and 28 only.

5 Right-click **Component 1 (comp1)>Rotating Machinery, Magnetic (rmm)>Ampère's Law 4** and choose **Rename**.

6 In the **Rename Ampère's Law** dialog box, type `Iron` in the **New label** text field.

7 Click **OK**.

Coil 1

1 In the **Model Builder** window, right-click **Rotating Machinery, Magnetic (rmm)** and choose the domain setting **Vector Potential>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 multi-turn**.
5 In the I_{coil} text field, type `0[A]`.
6 Locate the **Homogenized Multi-Turn Conductor** section. In the N text field, type `1`.
7 In the a_{coil} text field, type `A`.
8 Locate the **Coil** section. Select the **Coil group** check box.

Reversed Current Direction 1

1 Right-click **Component 1 (comp1)>Rotating Machinery, Magnetic (rmm)>Coil 1** and choose **Reversed Current Direction**.
2 Click the **Select Box** button on the **Graphics** toolbar.
3 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 1**).

Continuity 1

1 In the **Model Builder** window, right-click **Rotating Machinery, Magnetic (rmm)** and choose **Pairs>Continuity**.
2 In the **Settings** window for **Continuity**, locate the **Pair Selection** section.
3 In the **Pairs** list, select **Identity Boundary Pair 1 (ap1)**.

MESH 1

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Predefined** list, choose **Finer**.
4 Click the **Custom** button.
5 Locate the **Element Size Parameters** section. In the **Resolution of narrow regions** text field, type `2`.

6 Click **Build All**.

STUDY 1

Step 2: Time Dependent

1 On 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 **Times** text field, type `range(0,0.01,0.25)`.

Some adjustments to the solver settings are required in order to obtain a more stable solution.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

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

4 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Strict**.

5 Find the **Algebraic variable settings** subsection. From the **Error estimation** list, choose **Exclude algebraic**.

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

7 In the **Settings** window for **Fully Coupled**, click to expand the **Results while solving** section.

8 Click to expand the **Method and termination** section. Locate the **Method and Termination** section. From the **Jacobian update** list, choose **Once per time step**.

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

10 On the **Study** toolbar, click **Compute**.

RESULTS

Magnetic Flux Density (rmm)

1 In the **Model Builder** window, under **Results** click **Magnetic Flux Density (rmm)**.

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

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

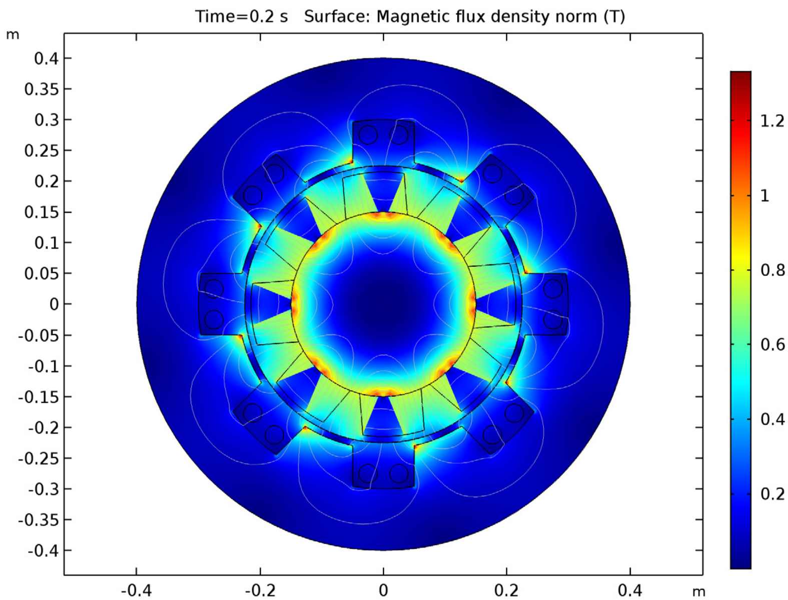
Contour 1

1 In the **Model Builder** window, expand the **Magnetic Flux Density (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 On the **Magnetic Flux Density (rmm)** toolbar, click **Plot**.



The plot will show 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.

1D Plot Group 2

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

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 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.

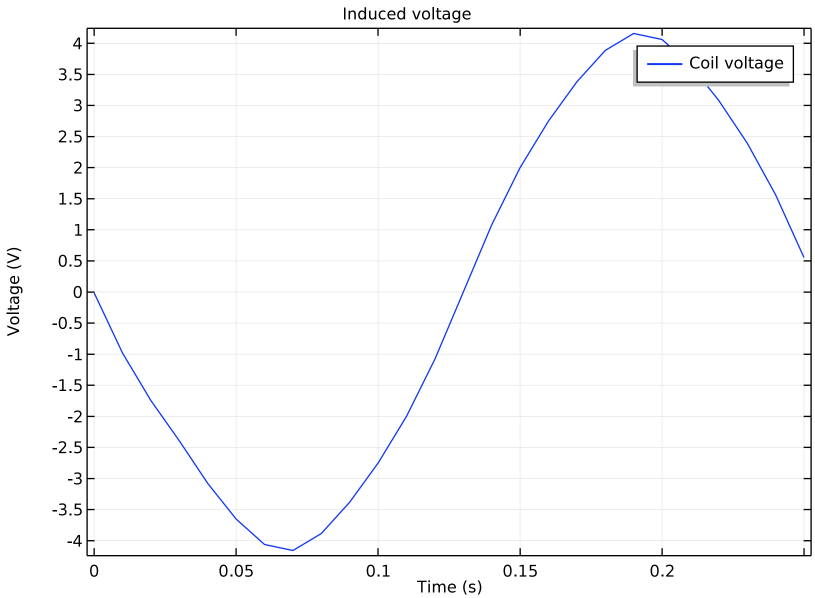
8 In the associated text field, type `Voltage (V)`.

Global 1

1 Right-click **1D Plot Group 2** 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>Rotating Machinery, Magnetic (Magnetic Fields)>Coil parameters>rmm.VCoil_1 - Coil voltage**.

3 On the **1D Plot Group 2** toolbar, click **Plot**.



The plot shows an induced EMF with an amplitude of about 4.2 V