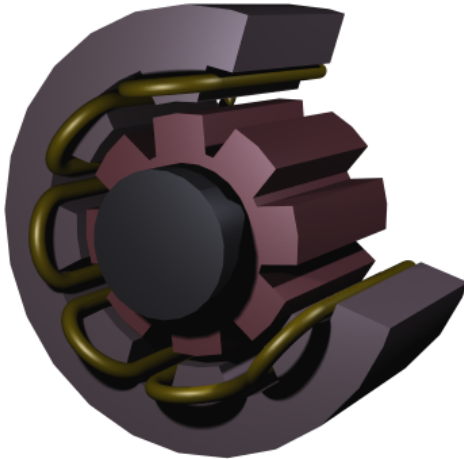# Generator in 2D

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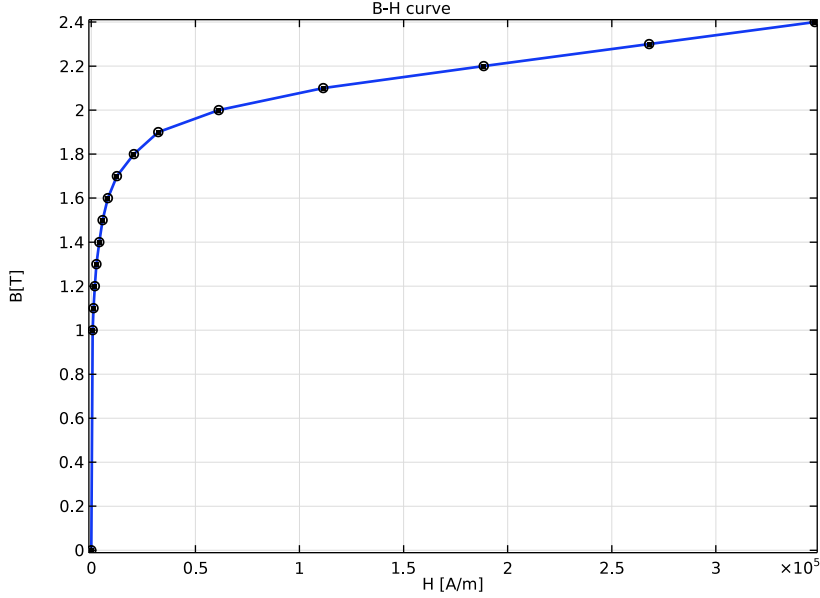
Figure 2: The norm of the magnetic flux, |$\mathbf{B}$|, versus the norm of the magnetic field, |$\mathbf{H}$|, for the rotor and stator materials.

The generated voltage is computed as the line integral of the electric field, $\mathbf{E}$, 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 $\mathbf{E}$ field for each winding cross-section, multiplying it by the axial length of the rotor, and taking the sum over all winding cross sections.

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

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.
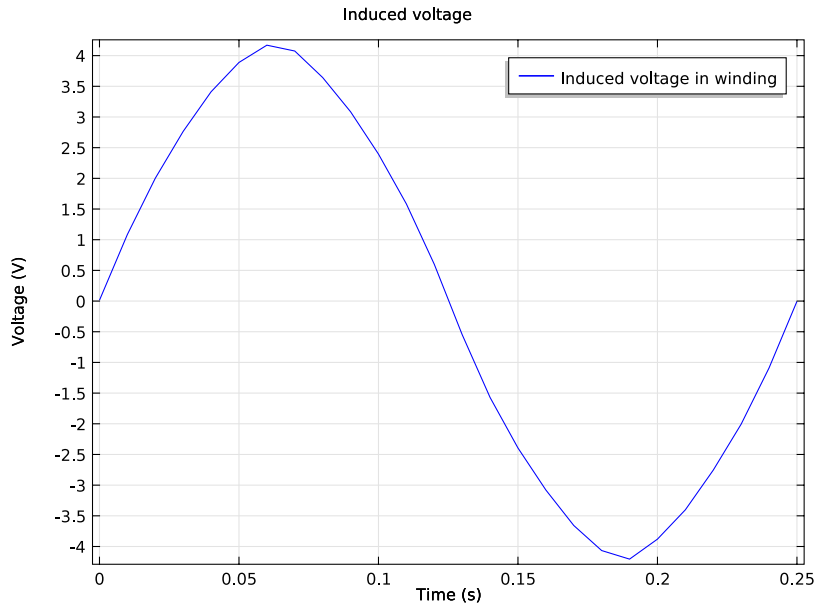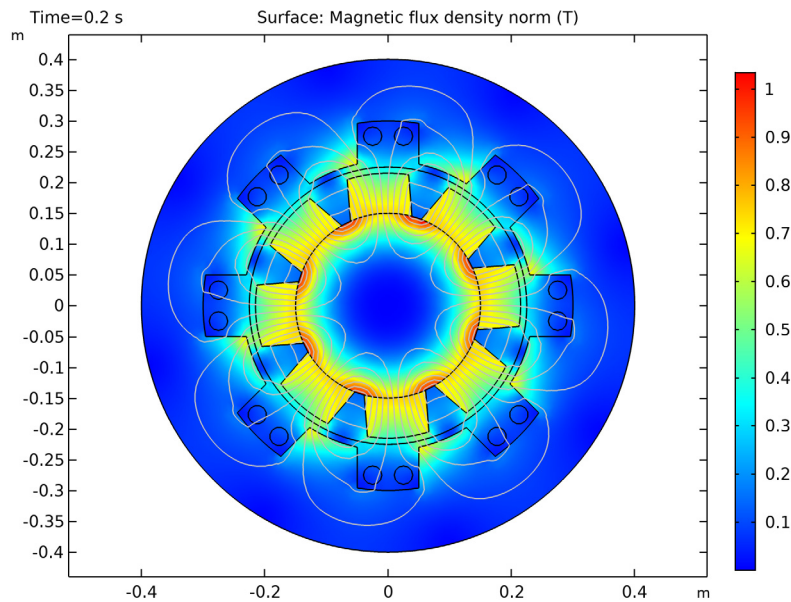

Figure 3: The generated voltage over one quarter of a revolution. This simulation used a single-turn winding.

The norm of the magnetic flux, |**B**|, and the field lines of the **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>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
A	pi*(3[cm]/2)^2	7.0686E-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 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 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**.

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

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.

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

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Recoil permeability	murec_iso ; murecii = murec_iso, murecij = 0	1	1	Remanent flux density
Remanent flux density norm	normBr	0.84[T]	T	Remanent flux density

5 In the **Label** text field, type `Generic SmCo Magnet`.

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.

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.

Ampère's Law 2

1 In the **Physics** toolbar, click **Domains** and choose **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 **Constitutive Relation B-H** section. From the **Magnetization model** list, choose **Remanent flux density**.

6 In the **Label** text field, type Permanent Magnets Outward.

The other four magnets are reversed.

Ampère's Law 3

1 In the **Physics** toolbar, click **Domains** and choose **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 **Constitutive Relation B-H** section. From the **Magnetization model** list, choose **Remanent flux density**.

6 Specify the **e** vector as

-1	r
0	phi
0	a

7 In the **Label** text field, type Permanent Magnets Inward.

Ampère's Law 4

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.

Coil 1

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 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 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 1**).

Continuity 1

1 In the **Physics** toolbar, in the **Boundary** section, 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 1 (ap1)** in the **Pairs** list.

5 Click **OK**.

STUDY 1

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

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

Size

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

2 From the **Predefined** list, choose **Coarser**.

3 Click the **Custom** button.

4 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.

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 **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.

Magnetic Flux Density (rmm)

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Results> 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**.

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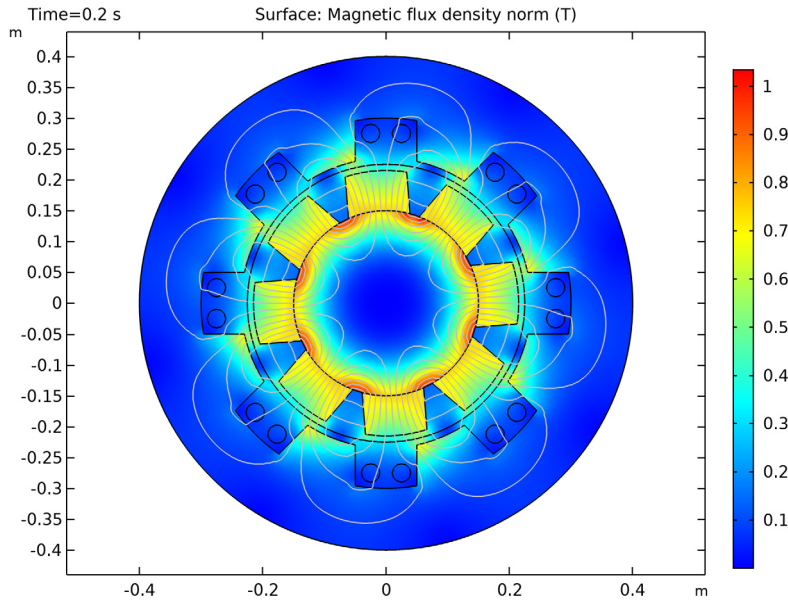

The plot will show the rotor position at $t = 0.2$ s, the magnetic flux density norm and magnetic flux lines. Next, plot the induced EMF in a quarter of the cycle.

1D Plot Group 2

1 In 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 - V**.

3 In the **1D Plot Group 2** toolbar, click **Plot**.
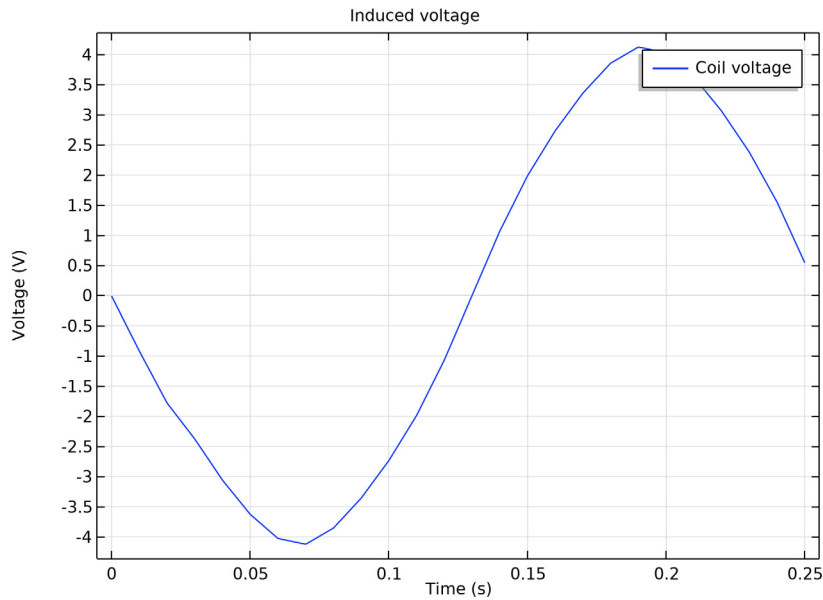
The plot shows an induced EMF with an amplitude of about 4.2 V