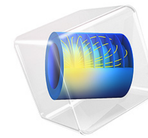
Created in COMSOL Multiphysics 6.3



Voltage Induced in a Coil by a Moving Magnet

Introduction

A magnet moving axially through the center of a coil induces a voltage across the coil terminals. A practical application of this phenomenon is in shaker flashlights, where the flashlight is vigorously shaken back and forth, thereby causing a magnet to move through a multiturn coil, which provides charge to the battery. This example models the motion of a magnet through a coil and computes the induced voltages. The displacement of the magnet is significant, so the application uses a moving and sliding mesh.

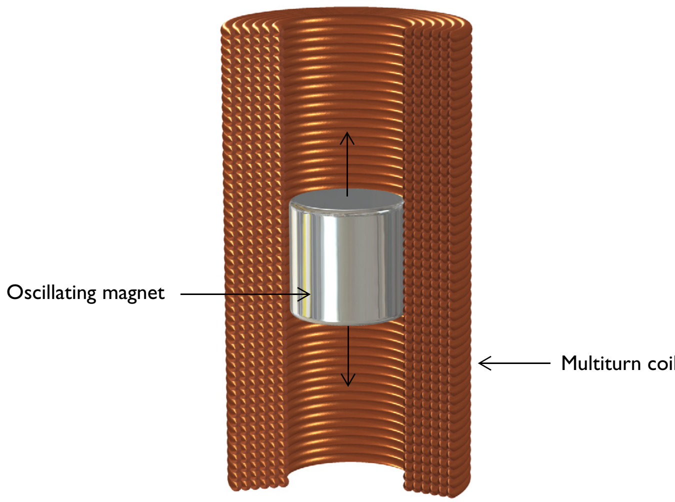


Figure 1: Drawing showing a sinusoidally oscillating magnet and a multiturn coil.

Model Definition

Figure 1 illustrates the system setup, in which a magnet of strength 1.2 T is displaced sinusoidally at 4 Hz with a peak displacement of 30 mm inside of an 800 turn coil. This results in a 2D axisymmetric problem, where the modeling space is a rectangular region in the *rz*-plane bounded by the Magnetic Insulation boundary condition, which represents a metallic enclosure.

Both the magnet and the multiturn coils are represented by rectangles. The magnet and coil corners are not rounded off, which leads to a simpler mesh and a smaller problem size. Although the sharp corners do introduce local singularities into the magnetic fields, it is not a concern for this type of application, whose single objective is to determine the induced voltage across the coil. This voltage is computed by taking the integral of the fields over the domains, which is quite insensitive to singularities in the fields. The corners of these domains only need to be rounded if forces and field strengths around the corners are of interest.

To define the displacement of the magnet and the surrounding air domains, the application uses the moving mesh functionality. Because neither the coil nor the air domain surrounding it needs to deform, it is sufficient to define the Moving Mesh interface in the magnet and the air domains above and below it.

When the domain movement is significant, it is warranted to use the sliding mesh functionality, introducing additional steps into the setup. When drawing the geometry, the Form Assembly functionality must be used to finalize the geometry. This feature assumes that all objects are disjoint, and automatically creates an Identity Pair at the touching boundaries between objects. The Identity Pair is used to define a Pair Continuity boundary condition in the Magnetic Fields interface, which specifies that the fields must be continuous across the noncongruent meshes. For higher accuracy, the application uses weak constraints and a smaller mesh size at these boundaries.

Solve the model in two steps. First, a stationary analysis of just the magnetic fields computes the fields due to the magnet at its starting location. This is needed to provide correct initial conditions for the subsequent transient analysis of the magnetic fields and the moving mesh. The tolerances are tightened slightly from their defaults.

Results and Discussion

Figure 2 shows the magnetic field and the mesh after 0.2 s, slightly less than the period of oscillation of the magnet, $T = 0.25$ s. The mesh is stretched and compressed in the air domains above and below the magnet. Although the mesh is noncongruent at the Identity Pair boundary, the Pair Continuity boundary condition ensures that the solution is continuous.

Figure 3 displays the induced voltage for the open circuit configuration of the coil.

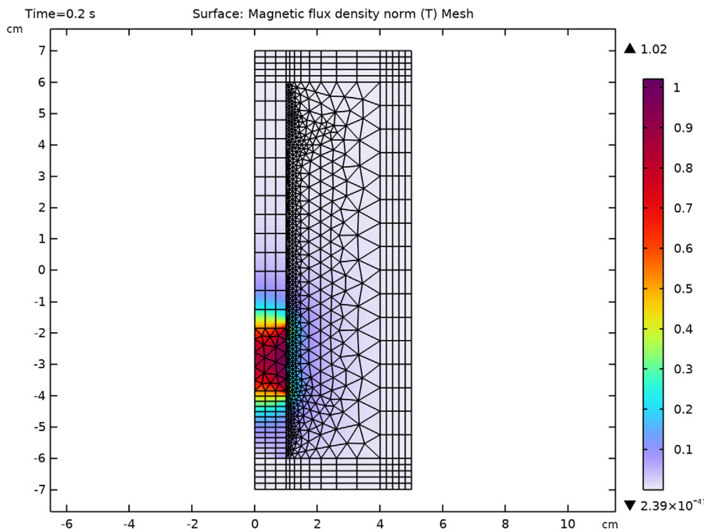


Figure 2: Magnetic flux density and deformed mesh at the bottom of the stroke of the magnet.

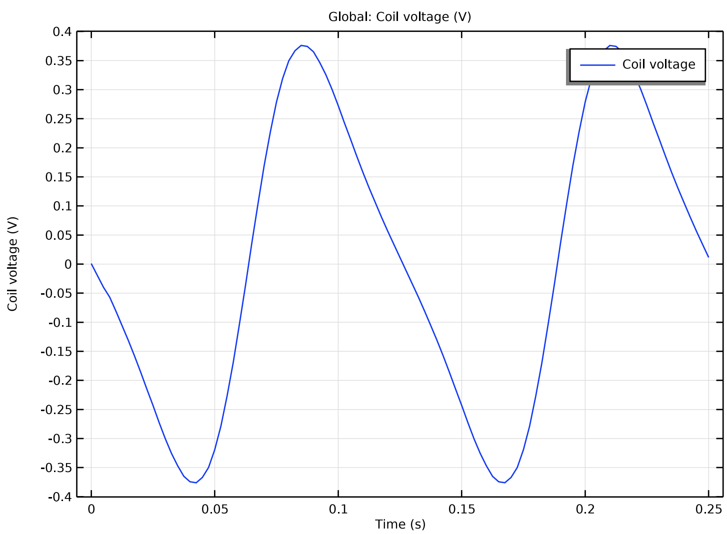


Figure 3: Induced voltage in the coil over time.

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

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 Axisymmetric**.
2 In the **Select Physics** tree, select **AC/DC** > **Electromagnetic Fields** > **Magnetic Fields (mf)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies** > **Stationary**.
6 Click **Done**.

GEOMETRY 1

Define the frequency of the oscillating magnet.

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
f0	4[Hz]	4 Hz	Frequency of an oscillating magnet
T0	1/f0	0.25 s	Time period of an oscillating magnet
t	0[s]	0 s	Time

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **cm**.

Follow these instructions to construct the model geometry.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Height** text field, type 2.

4 Locate the **Position** section. In the **z** text field, type -1.

5 Click **Build Selected**.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Height** text field, type 8.

4 Locate the **Position** section. In the **r** text field, type 1.1.

5 In the **z** text field, type -4.

6 Click **Build Selected**.

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

Rectangle 3 (r3)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Height** text field, type 12.

4 Locate the **Position** section. In the **z** text field, type -6.

5 Click **Build Selected**.

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

Rectangle 4 (r4)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 3.

4 In the **Height** text field, type 12.

5 Locate the **Position** section. In the **r** text field, type 1.

6 In the **z** text field, type -6.

Rectangle 5 (r5)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Position** section.
3 In the **z** text field, type 6.

Rectangle 6 (r6)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type 4.
4 Locate the **Position** section. In the **r** text field, type 1.
5 In the **z** text field, type 6.

Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.
2 Select the objects **r5** and **r6** only.
3 In the **Settings** window for **Mirror**, locate the **Normal Vector to Line of Reflection** section.
4 In the **r** text field, type 0.
5 In the **z** text field, type 1.
6 Locate the **Input** section. Select the **Keep input objects** checkbox.

Rectangle 7 (r7)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Height** text field, type 14.
4 Locate the **Position** section. In the **r** text field, type 4.
5 In the **z** text field, type -7.
6 Click **Build Selected**.

Form a union of **Rectangle 1** and **Rectangle 3**.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.
2 Select the objects **r1** and **r3** only.
3 In the **Settings** window for **Union**, click **Build Selected**.

Now, unify the rest of the geometry.

Union 2 (uni2)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **mir1(1)**, **mir1(2)**, **r2**, **r4**, **r5**, **r6**, and **r7** only.

3 In the **Settings** window for **Union**, click **Build Selected**.

Finish creating the geometry by using an assembly.

Form Union (fin)

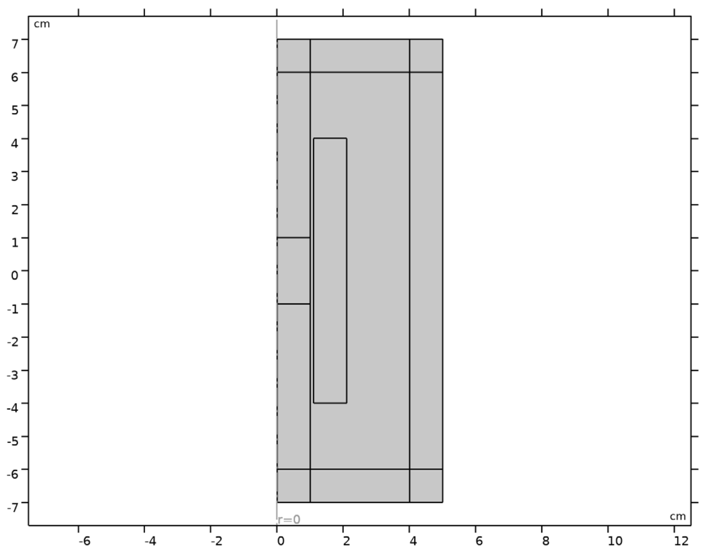
1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Form Union (fin)**.

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 Click **Build Selected**.

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



The final geometry is shown in the figure above.

Define boundary selections for the magnet and continuity pair.

DEFINITIONS

Magnet Boundaries

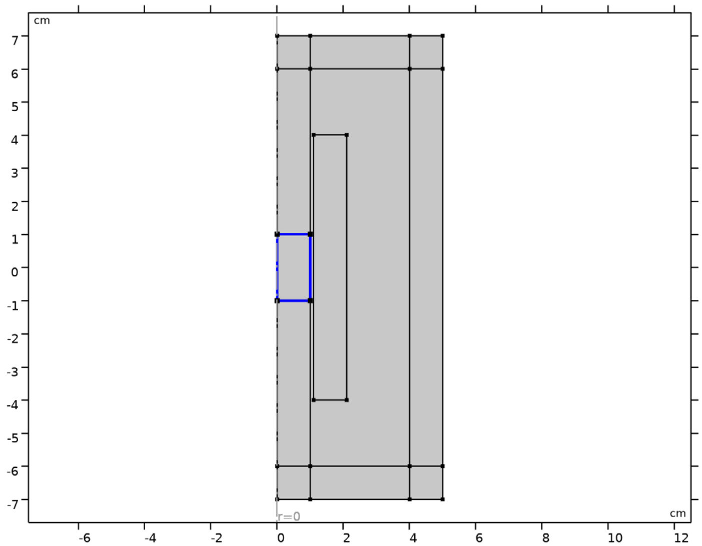
1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundaries 30, 31, 33, and 36 only.

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



Continuity Boundaries

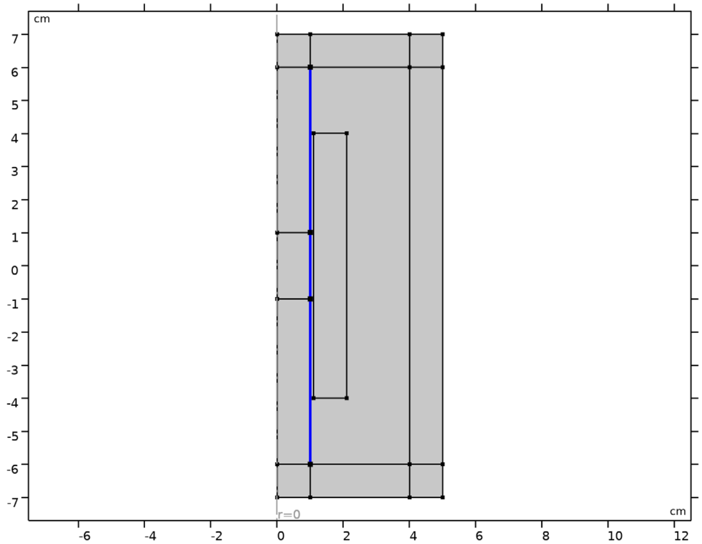
1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundaries 35–37 only.

5 In the **Label** text field, type `Continuity Boundaries`.



Next, add copper for the coil and NdFeB for the magnet.

ADD MATERIAL

1. In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **Built-in** > **Copper**.
4. Click the **Add to Component** button in the window toolbar.
5. In the tree, select **AC/DC** > **Magnetic Materials (Bomatec®)** > **NdFeB** > **BMN-35**.
6. Click the **Add to Component** button in the window toolbar.
7. In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Copper (mat1)

1. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
2. From the **Selection** list, choose **Manual**.
3. Click **Clear Selection**.
4. Select Domain 6 only.

BMN-35 (mat2)

1 In the **Model Builder** window, click **BMN-35 (mat2)**.

2 Select Domain 11 only.

Add the **Magnet** feature to model the moving magnet. The poles are defined on the top and bottom boundaries of the magnet.

MAGNETIC FIELDS (MF)

Magnet 1

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

2 Select Domain 11 only.

North 1

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

2 Select Boundary 33 only.

South 1

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

2 Select Boundary 31 only.

Next, add the **Coil** feature to model the pickup multiturn coil.

Coil 1

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

2 Select Domain 6 only.

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

4 From the **Material type** list, choose **Solid**.

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

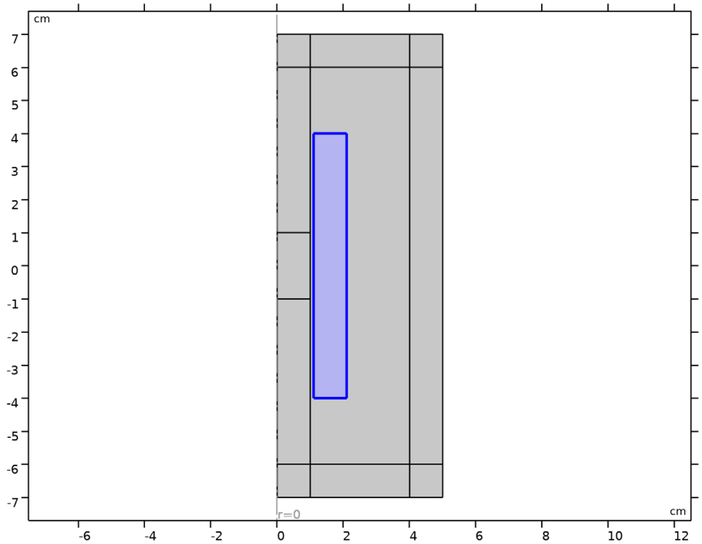
6 Locate the **Homogenized Conductor** section. In the N text field, type `800`.

7 From the list, choose **User defined**.

8 Find the **High-frequency effective loss** subsection. Clear the **Include harmonic loss** checkbox.

9 In the a text field, type `pi*(0.5[mm])^2`.

10 Locate the **Coil** section. In the I_{coil} text field, type 0.



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

12 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

13 Click **OK**.

Add a continuity boundary condition and enable the weak constraints features.

Continuity 1

In the **Physics** toolbar, click **Pairs** and choose **Continuity**.

Magnetic Insulation 1

1 In the **Model Builder** window, click **Magnetic Insulation 1**.

2 In the **Settings** window for **Magnetic Insulation**, click to expand the **Constraint Settings** section.

3 From the **Constraint** list, choose **Weak constraints**.

Continuity 1

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

2 In the **Settings** window for **Continuity**, locate the **Pair Selection** section.

3 Click **Add**.

4 In the **Add** dialog, select **Identity Boundary Pair 1 (ap1)** in the **Pairs** list.

5 Click **OK**.

6 In the **Settings** window for **Continuity**, click to expand the **Constraint Settings** section.

7 From the **Constraint** list, choose **Weak constraints**.

Define the **Moving Mesh** only in the domains to the left of the continuity pair.

COMPONENT 1 (COMP1)

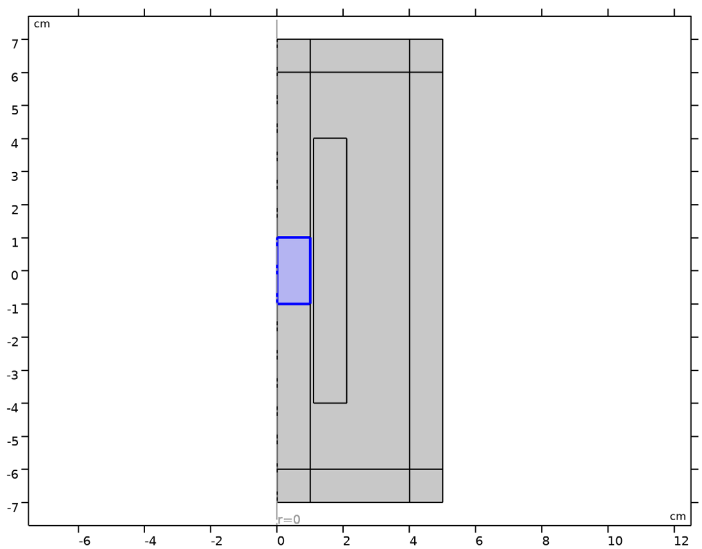
Prescribed Deformation 1

1 In the **Model Builder** window, expand the **Continuity 1** node.

2 Right-click **Component 1 (comp1)** and choose **Moving Mesh** > **Prescribed Deformation**.

3 Select Domain 11 only.

4 In the **Settings** window for **Prescribed Deformation**, locate the **Prescribed Deformation** section.



5 Specify the *dx* vector as

0	R
30[mm]*sin(2*pi*f0*t)	Z

Prescribed Deformation 2

1 In the **Moving Mesh** toolbar, click **Prescribed Deformation**.

2 Select Domain 12 only.

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

4 Specify the dx vector as

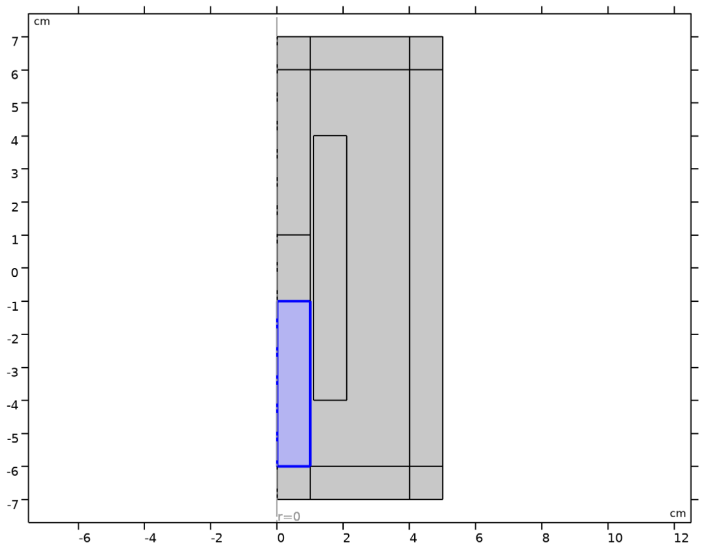
0	R
30[mm]*sin(2*pi*f0*t)*(6[cm]-Z)/5[cm]	Z

Prescribed Deformation 3

1 In the **Moving Mesh** toolbar, click **Prescribed Deformation**.

2 Select Domain 10 only.

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



4 Specify the dx vector as

0	R
30[mm]*sin(2*pi*f0*t)*(6[cm]+Z)/5[cm]	Z

Define the **Infinite Element Domain**.

DEFINITIONS

Infinite Element Domain 1 (ie1)

1 In the **Definitions** toolbar, click **Infinite Element Domain**.

2 In the **Settings** window for **Infinite Element Domain**, locate the **Geometry** section.

3 From the **Type** list, choose **Cylindrical**.

4 Select Domains 1–3, 5, and 7–9 only.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

Size 1

1 Right-click **Component 1 (comp1)** > **Mesh 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 Select Boundaries 28, 29, and 31–37 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `4[mm]`.

Size 2

1 In the **Model Builder** window, right-click **Mesh 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 Select Boundary 9 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `1[mm]`.

Edge 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.

2 Select Boundaries 28, 31–35, and 37 only.

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

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

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

4 Select Domains 10 and 12 only.

Free Triangular 1

1. In the **Mesh** toolbar, click **Free Triangular**.
2. In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 4, 6, and 11 only.

Copy Edge 1

1. Right-click **Mesh 1** and choose **Copying Operations** > **Copy Edge**.
2. Select Boundaries 12 and 34 only.
3. In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.
4. Click to select the **Activate Selection** toggle button.
5. Select Boundaries 6 and 13 only.

Copy Edge 2

1. Right-click **Mesh 1** and choose **Copying Operations** > **Copy Edge**.
2. Select Boundaries 10 and 29 only.
3. In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.
4. Click to select the **Activate Selection** toggle button.
5. Select Boundaries 2 and 8 only.

Edge 2

1. In the **Mesh** toolbar, click **More Generators** and choose **Edge**.
2. Select Boundaries 1, 4, 18, 19, 21–25, and 27 only.

Distribution 1

Right-click **Edge 2** and choose **Distribution**.

Copy Edge 3

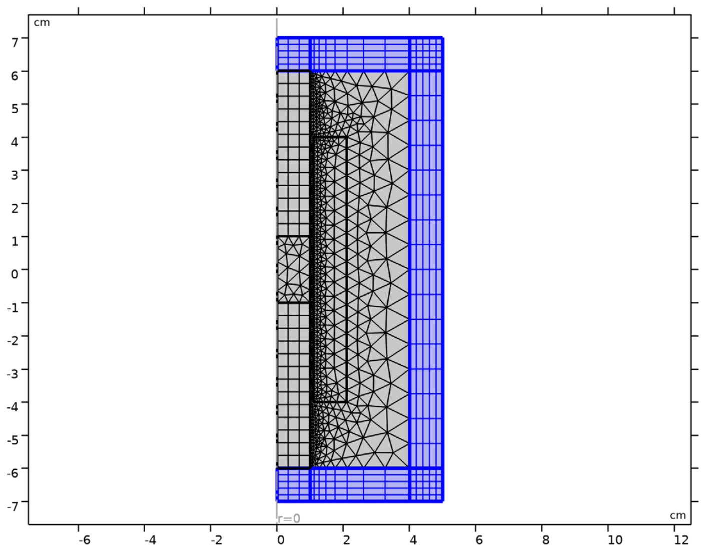
1. In the **Model Builder** window, right-click **Mesh 1** and choose **Copying Operations** > **Copy Edge**.
2. In the **Settings** window for **Copy Edge**, locate the **Source Boundaries** section.
3. Click **Remove from Selection**.
4. Select Boundaries 1 and 4 only.
5. Locate the **Destination Boundaries** section. Click to select the **Activate Selection** toggle button.
6. Select Boundaries 7 and 11 only.

Mapped 2

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, click **Build All**.

The mesh should look like that shown in the figure below.



Now, add the **Time Dependent** study step and solve the problem in the time domain. The **Time Dependent** study automatically takes the initial values for the vector potential from the stationary solution. Some tuning of the solver is done in order to speedup the calculation.

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,T0/100,T0)`.

Use the following steps to improve convergence speed and robustness.

Solution 1 (sol1)

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

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Study Settings** section.

3 From the **Tolerance** list, choose **User controlled**.

4 In the **Relative tolerance** text field, type `1e-4`.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

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

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

4 From the **Jacobian update** list, choose **On every iteration**.

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

RESULTS

Magnetic Flux Density (mf)

Use the following steps to generate a plot of the magnetic flux density norm and deformed mesh as shown in Figure 2.

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 **Model Builder** window, expand the **Magnetic Flux Density (mf)** node.

Contour 1, Streamline 1

1 In the **Model Builder** window, under **Results** > **Magnetic Flux Density (mf)**, Ctrl-click to select **Streamline 1** and **Contour 1**.

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

Mesh 1

1 In the **Model Builder** window, right-click **Magnetic Flux Density (mf)** and choose **Mesh**.

2 In the **Settings** window for **Mesh**, locate the **Coloring and Style** section.

3 From the **Element color** list, choose **None**.

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

Finally, reproduce the plot for an induced voltage in the coil.

1D Plot Group 4

In the **Results** toolbar, click **1D Plot Group**.

Global 1

1 Right-click **1D Plot Group 4** 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)** > **Magnetic Fields** > **Coil parameters** > **mf.VCoil_1 - Coil voltage - V**.

3 In the **1D Plot Group 4** toolbar, click **Plot**.

Coil Induced Voltage

1 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Rename**.

2 In the **Rename 1D Plot Group** dialog, type `Coil Induced Voltage` in the **New label** text field.

3 Click **OK**.

Compare the plot with Figure 3.