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 multi-turn 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.

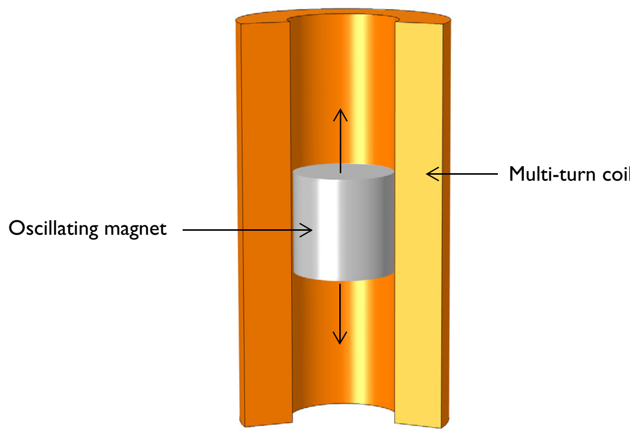


Figure 1: Drawing showing a sinusoidally oscillating magnet and a multi-turn coil.

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 multi-turn coils are represented by rectangles. The magnet and coil corners are not rounded off, which leads to a simpler mesh and 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 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.

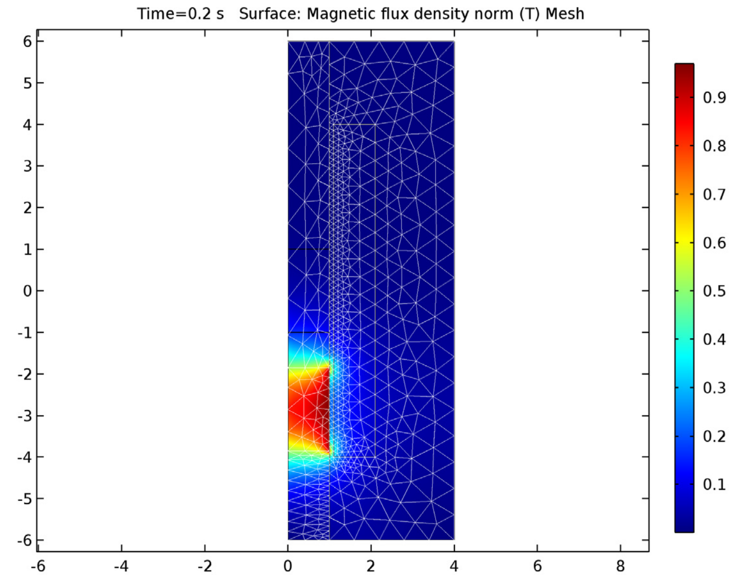


Figure 2: Magnetic flux density and deformed mesh at the bottom of the stroke of the magnet.

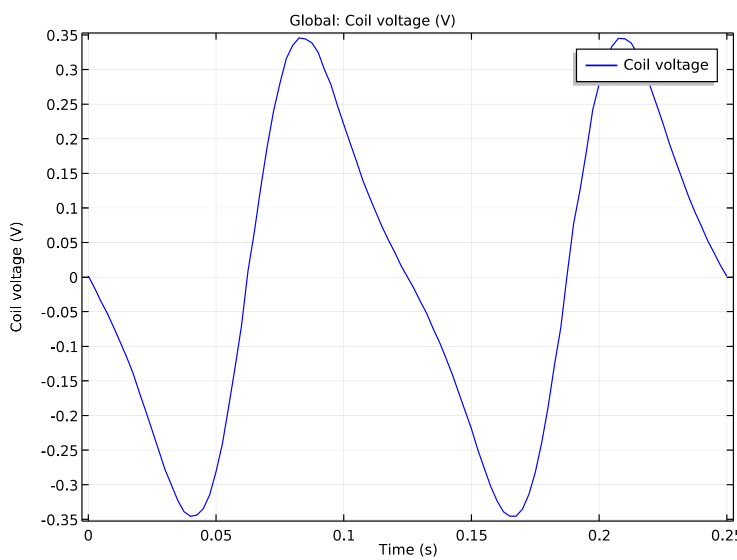


Figure 3: Induced voltage in the coil over time.

Application Library path: `ACDC_Module/Motors_and_Actuators/induced_voltage_moving_magnet`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

1 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>Magnetic Fields (mf)**.

3 Click **Add**.

4 In the **Select physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh (ale)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

8 Click **Done**.

Define the frequency of a oscillating magnet.

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

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 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

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

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

5 Right-click **Component 1 (comp1)>Geometry 1>Rectangle 1 (r1)** and choose **Build Selected**.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size** 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 Right-click **Component 1 (comp1)>Geometry 1>Rectangle 2 (r2)** and choose **Build Selected**.

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

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

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

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

5 Right-click **Component 1 (comp1)>Geometry 1>Rectangle 3 (r3)** and choose **Build Selected**.

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

Rectangle 4 (r4)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size** 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`.
7 Right-click **Component 1 (comp1)>Geometry 1>Rectangle 4 (r4)** and choose **Build Selected**.

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

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.
2 Select the objects **r1** and **r3** only.
3 Right-click **Component 1 (comp1)>Geometry 1>Union 1 (uni1)** and choose **Build Selected**.

Next, form a union of a **Rectangle 2** and **Rectangle 4**.

Union 2 (uni2)

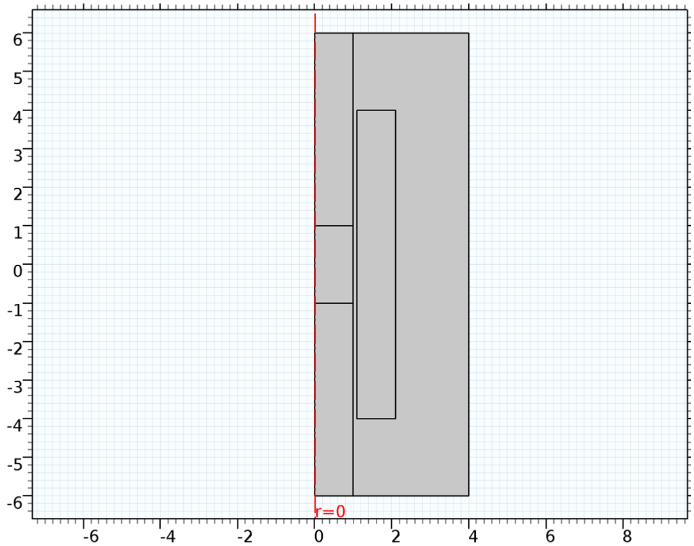
1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.
2 Select the objects **r4** and **r2** only.
3 Right-click **Component 1 (comp1)>Geometry 1>Union 2 (uni2)** and choose **Build Selected**.

Finish the geometry creation 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 In the **Relative repair tolerance** text field, type `1e-6`.
4 From the **Action** list, choose **Form an assembly**.

5 Right-click **Component 1 (comp1)>Geometry 1>Form Union (fin)** and choose **Build Selected**.



The final geometry looks as shown in the figure above.

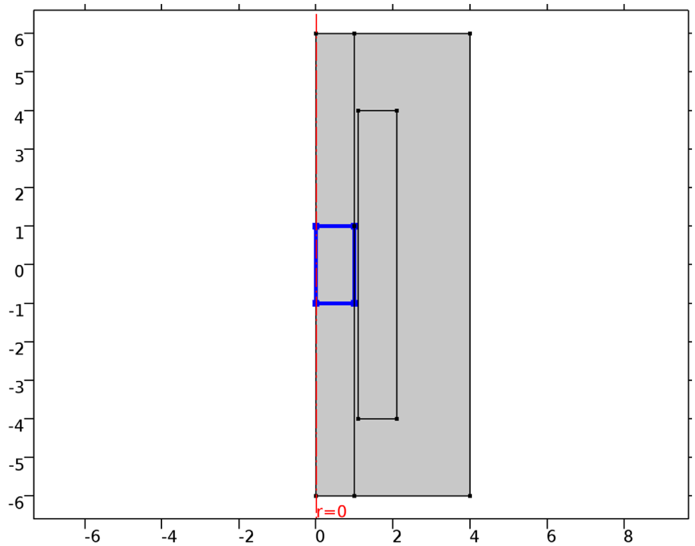
DEFINITIONS

Define boundary selections for the magnet and continuity pair.

Explicit 1

1. On 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 3, 4, 6, and 9 only.
5. Right-click **Component 1 (comp1)>Definitions>Explicit 1** and choose **Rename**.
6. In the **Rename Explicit** dialog box, type `Magnet Boundaries` in the **New label** text field.

7 Click **OK**.



Explicit 2

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

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

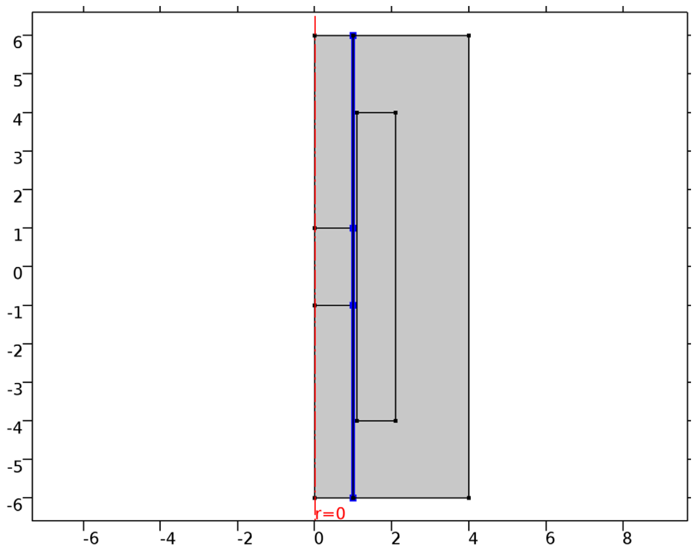
3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 8–10 only.

5 Right-click **Component 1 (comp1)>Definitions>Explicit 2** and choose **Rename**.

6 In the **Rename Explicit** dialog box, type `Continuity Boundaries` in the **New label** text field.

7 Click **OK**.



Now, assign air for all the domains.

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

MAGNETIC FIELDS (MF)

Now set up the physics for the magnetic field. Apply Ampère's law in the magnet and the air domain.

Ampère's Law 2

1. On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.
2. Select Domain 2 only.
3. In the **Settings** window for Ampère's Law, locate the **Magnetic Field** section.
4. From the **Constitutive relation** list, choose **Remanent flux density**.

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

0	r
0	phi
1.2	z

Next, add the Multi-Turn Coil feature to model the coil.

Multi-Turn Coil 1

1 On the **Physics** toolbar, click **Domains** and choose **Multi-Turn Coil**.

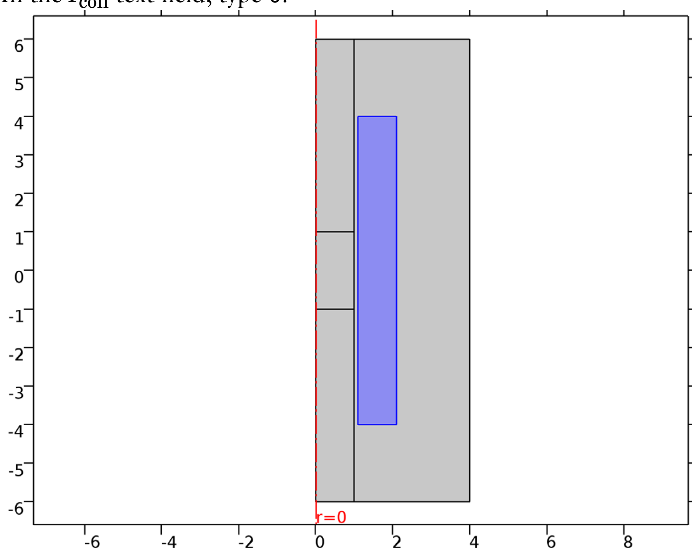
2 Select Domain 5 only.

3 In the **Settings** window for Multi-Turn Coil, locate the **Multi-Turn Coil** section.

4 Find the **Coil parameters** subsection. In the N text field, type `800`.

5 In the a_{coil} text field, type `pi*(0.5[mm])^2`.

6 In the I_{coil} text field, type `0`.



7 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

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

Continuity 1

1 On 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 In the **Pairs** list, select **Identity Pair 1 (ap1)**.

4 Click to expand the **Constraint settings** section. Locate the **Constraint Settings** section. Select the **Use weak constraints** check box.

Magnetic Insulation 1

1 In the **Model Builder** window, expand the **Continuity 1** node, then click **Magnetic Insulation 1**.

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

3 Locate the **Constraint Settings** section. Select the **Use weak constraints** check box.

Use the Moving Mesh interface only in the domains to the left of the continuity pair.

MOVING MESH (ALE)

On the **Physics** toolbar, click **Magnetic Fields (mf)** and choose **Moving Mesh (ale)**.

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

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

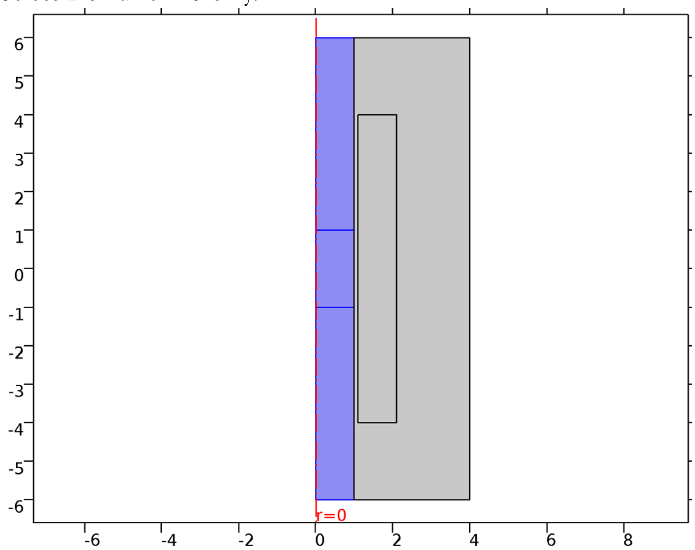
3 From the **Selection** list, choose **Manual**.

4 Select Domains 1–3 only.

Free Deformation 1

1 On the **Physics** toolbar, click **Domains** and choose **Free Deformation**.

2 Select Domains 1–3 only.



Prescribed Mesh Displacement 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 In the **Settings** window for Prescribed Mesh Displacement, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Magnet Boundaries**.

4 Locate the **Prescribed Mesh Displacement** section. In the d_z text field, type `30[mm]*sin(2*pi*f0*t)`.

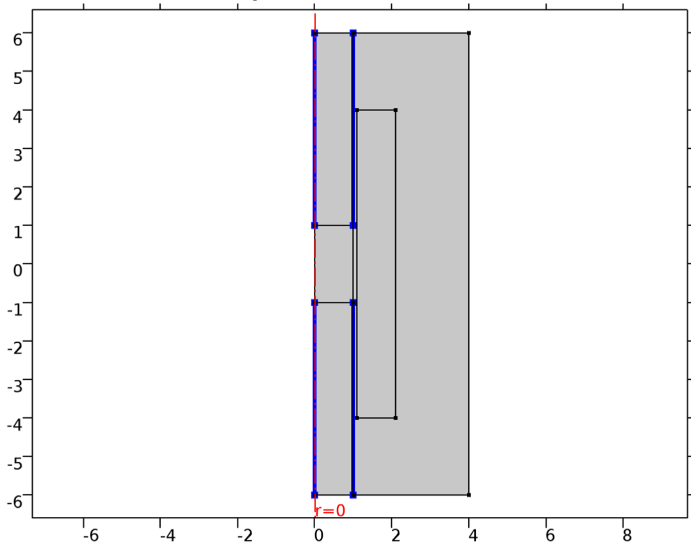
Prescribed Mesh Displacement 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 Select Boundaries 1, 5, 8, and 10 only.

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

4 Clear the **Prescribed z displacement** check box.



MESH 1

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

2 In the **Settings** window for Mesh, locate the **Mesh Settings** 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 From the **Selection** list, choose **Continuity Boundaries**.

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

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

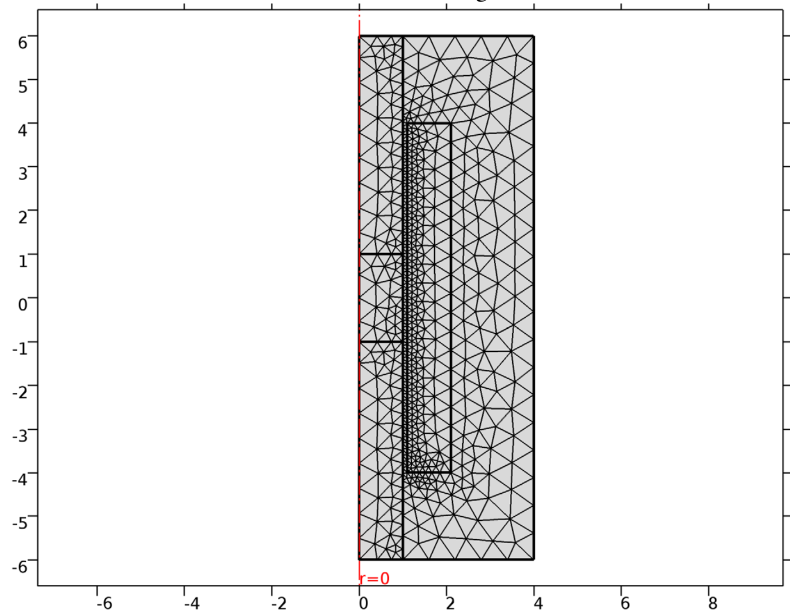
7 In the associated text field, type `3[mm]`.

Free Triangular 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Triangular**.

2 Right-click **Free Triangular 1** and choose **Build All**.

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



STUDY 1

Step 1: Stationary

Solve for the Magnetic Fields physics only in the stationary case.

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

2 In the **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Moving Mesh		physics

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.

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,T0/100,T0)`.

4 Select the **Relative tolerance** check box.

5 In the associated text field, type `0.0001`.

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

RESULTS

Magnetic Flux Density Norm (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 Right-click **Results>Magnetic Flux Density Norm (mf)** and choose **Mesh**.

4 In the **Settings** window for Mesh, locate the **Color** section.

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

6 From the **Wireframe color** list, choose **White**.

7 On the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

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

1D Plot Group 3

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

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

3 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>Magnetic Fields>Coil parameters>mf.VCoil_1 - Coil voltage**.

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

5 In the **Model Builder** window, right-click **1D Plot Group 3** and choose **Rename**.

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

7 Click **OK**.

Compare the plot with Figure 3